PowerLogic User's Guide

Version 3.0

PADS Software, Inc.
165 Forest Street
Marlborough, MA 01752
U.S.A.

Chapter 1
The PowerLogic Interface

The following topics guide you through the basics of starting and using the PowerLogic interface. Topics include:

Starting PowerLogic
Using the Interface
Menu Commands
Toolbars and Toolboxes
Work Area and Grid Settings
Controlling the View
Modeless Commands
Shortcut Keys
Pop-Up Menus

Starting PowerLogic

To start PowerLogic, double-click on the PowerLogic icon in the PowerLogic window. You can also choose Start/Programs/PADS-PowerLogic/PowerLogic.

Start-up Options

Additional start-up options are available, called command line switches. Command line switches allow you to enable different options, to open a file, start macros, and record your PowerLogic session. You can start the following command line options when you start PowerLogic:

Note: Options are not case-sensitive unless otherwise specified.

- /BMW

  Creates media from the previous session each time you start PowerLogic. /bmw is the Media Wizard. Using this command switch creates a directory and places necessary files in it to simplify the process of reproducing a PowerLogic problematic session. This is also a modeless command. Using the modeless command, you can also create media of the current session. To use the BMW command line switch, type /BMW or /BMWxx, where xx is your initials, in the command line. Note capitalization.

  This option is associated with another modeless command, BLT. BLT is the Log Test; it finds and runs the BMW macros to playback a recorded PowerLogic session. See the "Modeless Commands" topic in this chapter for more information.

- /build

  Includes the build number in the Help/About option.

- filename

  Opens the specified filename when opening PowerLogic. You must include the path to the file. Type the filename. Do not use a hyphen before the filename in the command line.

- /l

  Opens the last file you had open when you start PowerLogic.

- /mmacro name

  Runs the specified macro in the default macro file; for example, to run the macro file MyMacro, type **/mMyMacro**.

- /Mmacro file

  Specifies what file to use as the default macro file; for example, to run the macro MyMacro that lives in the macro file user1.mcr, type **/Muser1.mcr /mMyMacro**.

- /nc

  Starts PowerLogic without displaying the splash screen that includes copyright information.

Using Start-up Options

To use command line options:

1. Choose Start/Settings/Taskbar.
2. Choose the Start Menu Programs tab and choose the Advanced button.
3. Highlight the icon in which to add the command line option.
4. Choose File/Properties. Choose the Shortcut tab.
5. At the end of the executable string in the Target text box, press the Spacebar and type the command line switch you want to use. For example, to start a macro in the default macro file, the command line should read: c:\padspwr\powerlogic\powerlogic.exe /mMyMacro.

Note: When specifying a file to start, do not use / before the filename.

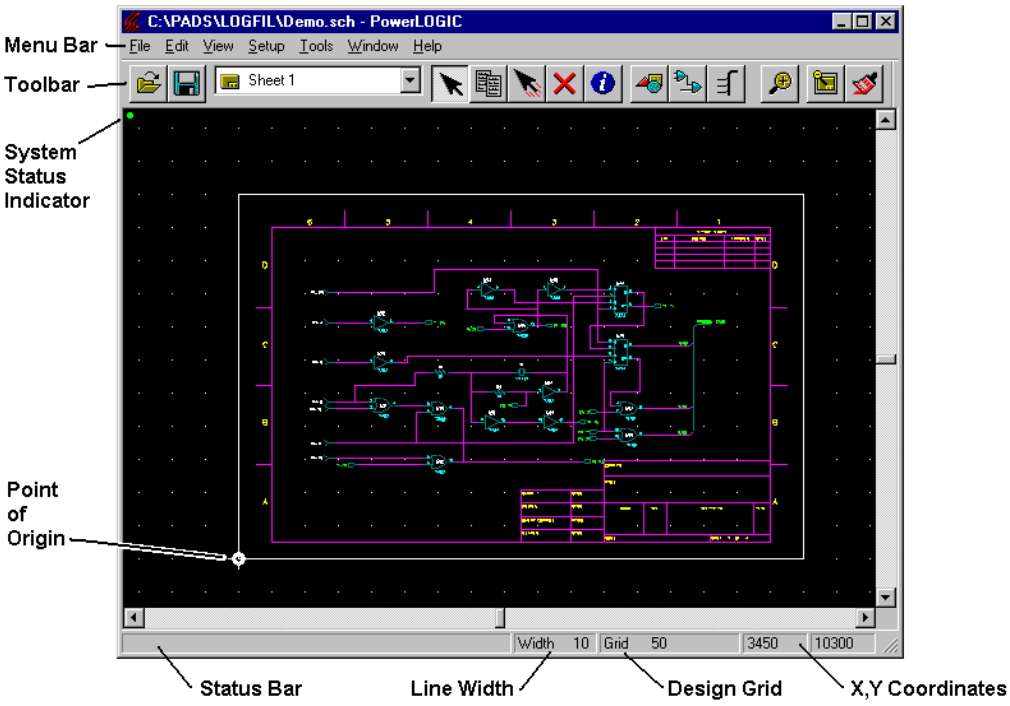
You can specify multiple command line switches.

You can also add a command line option to a shortcut icon:

1. Before starting PowerLogic, create a shortcut to PowerLogic.
2. Highlight the PowerLogic shortcut icon.
3. Click the right mouse button and choose Properties. Choose the Shortcut tab.
4. Follow Step 5 above.
5. Choose OK to close the Properties dialog box and start PowerLogic.

Using the Interface

PowerLogic uses standard Windows menus and icons to access commands.



Menu Bar

Lists the PowerLogic commands in pull-down menus.

Toolbar

Contains icons for opening and saving designs, sheet selection, zoom mode, redraws, and access to the toolboxes. See the "Toolbars and Toolboxes" topic in this chapter for additional information.

System Status Indicator

Indicates the processing state of PowerLogic. Red indicates that PowerLogic is processing and cannot receive input. Green indicates an idle or available state.

Point of Origin

Displays the 0,0 coordinate location. X,Y coordinates are calculated from this point.

Status Bar

Displays information on a selected connection, route, or component.

Line Width

Displays the default line width setting.

Design Grid

Displays the design grid setting.

X,Y Coordinates

Displays the horizontal distance of the cursor from 0,0 as the X coordinate. Displays the vertical distance of the cursor from 0,0 as the Y coordinate.

Work Area

This is the area where all design information is displayed. Shows the origin (0,0 coordinate) marker and display grid.

The display grid is a drafting aid, set in the Preferences dialog box using the Global tab.

Menu Commands

File Menu

New

Clears the current design file (if one is loaded) or erases any existing objects. If data exists, a warning window appears to confirm the action.

Open

Opens the File Open dialog box.

Save

Saves the changes to the current schematic. If this is a new, unnamed schematic, the File Save As dialog box appears.

Save As

Opens the File Save As dialog box and automatically places the name of the currently open file in the filename box. You can save an existing file with a new name or to a different directory using Save As.

Import

Opens the File Import dialog box.

Export

Opens the File Export dialog box.

Library

Opens the Library Manager dialog box.

Report

Opens the Reports dialog box.

Print Setup

Accesses the Windows Print Setup dialog box.

Print/Plot

Opens the Print/Plot dialog box.

Exit

Exits PowerLogic. If data exists, a warning window appears to confirm the action.

Loaded File List

Lists the last loaded files, up to four, at the bottom of the File menu.

Edit Menu

Undo

Rescinds the last editing action such as move, delete, swap, add connection, create 2D line, and others.

Redo

Repeats the last editing action made prior to an undo action.

Cut

Removes a group from the design and places the group into a temporary .grp file. PowerLogic adds an off-page reference symbol at the break point for connections that extend outside the group.

Copy

Copies a group into a temporary .grp file. PowerLogic adds an off-page reference symbol at the break point for connections that extend outside the group.

Paste

Places the contents of the temporary .grp file where you indicate. Also, you can close the design and open a different design to paste into. You can repeatedly paste the contents of the temporary .grp file until you overwrite the buffer with another cut or copy action.

Copy As Bitmap

Lets you define a rectangular area to copy graphics information to the Windows Clipboard as a bitmap image. Once copied, you can, for example switch to Microsoft Word and use the Edit/Paste command to insert the bitmap into a document.

To create a bitmap:

1. Choose Edit/Copy as Bitmap.
2. Area-select the objects to copy.

PowerLogic copies all items in the rectangle including the background, dot grid, color, etc. Open the application to place the bitmap and use the paste command to place the image.

Note: In addition to generating a bitmap of the defined area, PowerLogic copies the entire work area as a picture that you can insert with the Paste Special command in products that support this feature.

Copy to File

Copies a group of selected objects to a file.

1. In Autoselect mode, group-select the objects.
2. Choose Edit/Copy to File from the menu.
3. Type a path and filename in the Group Save File dialog box. Groups are saved into files with a .grp extension.
4. Choose Save.

Paste from File

Pastes the file contents into the current schematic.

1. Choose Edit/Paste from File from the menu.
2. Choose the file in the Load Group File dialog box.
3. Indicate the location.
4. Choose Open.

Move

Moves a group of selected objects.

1. In Autoselect mode, group-select the objects.
2. Choose Edit/Move from the menu.
3. Indicate the location.

Delete

Deletes a group of selected objects.

1. In Autoselect mode, group-select the objects.
2. Choose Edit/Delete from the menu.
3. Confirm the deletion.

Group Copy

Copies a group of selected objects.

1. In Autoselect mode, group-select the objects.
2. Choose Edit/Group Copy from the menu.
3. Indicate the location.

Filter

Sets one of the selection filters or enters Group Selection mode.

Select Group	Places PowerLogic in Group Selection Mode.
Select Parts	Turns on the Part Filter. See the "Object Selector Toolbar" topic in this chapter for more information.
Select Gates	Turns on the Gates Filter. See the "Object Selector Toolbar" topic in this chapter for more information.
Select Nets	Turns on the Net Filter. See the "Object Selector Toolbar" topic in this chapter for more information.
Select Pins	Turns on the Pin Filter. See the "Object Selector Toolbar" topic in this chapter for more information.

Insert PowerPCB Object

Links to a PowerPCB file or embeds a PowerPCB object into a PowerLogic design. See the "OLE in PowerLogic" chapter for more information.

Insert New Object

Opens the Insert Object dialog box where you can insert a linked or embedded object in PowerLogic. See the "OLE in PowerLogic" chapter for more information.

Delete All OLE Objects

Allows you to remove all of the OLE objects in a design. The prompt, *All current OLE Linked and Embedded objects will be removed from this Design. Do you want to continue?*, appears.

Links

Opens the Links dialog box where you can edit the link of an OLE object. See the "Editing OLE Links" topic in the "OLE in PowerLogic" chapter for more information.

Object

Opens, edits, converts your object. This menu changes depending on the OLE object. See the "OLE in PowerLogic" chapter for more information.

View Menu

Zoom

Choosing the Zoom icon invokes Zoom mode. The magnifying glass cursor indicates Zoom mode. To zoom in, locate the cursor to indicate the center of the zoom area and press the left button. To zoom out, locate the cursor and press the right button.

You can specify the zoom area. Press and hold the left button and drag diagonally upward to zoom in, diagonally downward to zoom out.

Sheet

Adjusts the display to fit the sheet outline in the work area. Information that is external to the sheet outline, such as dimensions and notes, does not display.

Redraw

Refreshes the work area display without modifying the current zoom factor or coordinate position.

Extents

Fits the display to encompass all objects contained in the current design. This includes objects contained on a non-viewed layer.

Push Hierarchy

Pushes down the hierarchy into a hierarchical symbol to view the underlying logic.

Pop Hierarchy

Pops up the hierarchy to the next level.

Capture

Opens the Capture View dialog box for saving and restoring work area views.

Previous View

Restores the previous view of the work area.

Setup Menu

Sheets

Opens the Sheets dialog box.

Preferences

Opens the Preferences dialog box.

Design Rules

Opens the Rules dialog box.

Layer Definition

Opens the Layers Setup dialog box.

Display Colors

Opens the Display Colors dialog box.

Tools Menu

Part Editor

Starts the Part Editor, disabling the PowerLogic Schematic Editor.

Attribute Manager

Displays the Manage Schematic Attributes dialog box where you can list the attribute names in your schematic and to add an attribute to, rename an attribute in, or delete an attribute from the schematic. See the "Library Operations" chapter for more information.

Update OffPage from Library

Opens the Update Off Page from Library dialog box.

Update Pins from Library

Opens the Update Pins from Library dialog box.

Save OffPage to Library

Opens the Save Off Page to Library dialog box.

Netlist to PCB

Opens the Netlist dialog box.

Forward ECO to PCB

Opens the Forward ECO to PCB dialog box.

Backward Annotate from PCB

Opens the Backward Annotate from PCB dialog box.

Export Rules to PCB

Opens the Export Rules to PCB ASCII File dialog box.

Import Rules from PCB

Opens the Import Rules from PCB ASCII File dialog box.

OLE PowerPCB Connection

Opens the Connect to PowerPCB dialog box.

Visual Basic Scripting

Opens the Visual Basic Editor and Visual Basic Scripts dialog boxes where you can develop Visual Basic scripts or load your most frequently used scripts and run them independently. See the "Visual Basic Scripting" topic in the "OLE in PowerLogic" chapter for more information.

Macros

Opens the Macro dialog box.

Window Menu

Menu Bar

Turns the menu bar display on and off. Ctrl-Alt-M also toggles the menu display.

Toolbars

Turns the toolbar display on and off.

Status Bar

Turns the message line display at the bottom of the screen on and off.

Drafting Toolbox

Turns the Drafting toolbox display on or off.

Design Toolbox

Turns the Design toolbox display on or off.

Bus Toolbox

Turns the Bus toolbox display on or off.

Status

Turns the Status Window display on and off.

Help Menu

Contents

Displays the On-line Help Contents page.

Search

Starts the keyword search utility for PowerLogic's On-line Help.

How To Use

Displays instructions for using Help.

Modeless Commands

Opens the On-line Help "Modeless Commands" topic.

Frequently Asked Questions

Opens the On-line Help "Frequently Asked Questions" topic.

About

Displays PowerLogic version and user information.

Toolbars and Toolboxes

Toolbars

PowerLogic has two available toolbars: the Standard toolbar and the Object Selector toolbar.

The Standard toolbar is the row of icons under the menu bar. This toolbar changes when you invoke or exit the Part Editor.

The Object Selector toolbar appears at the bottom of the PowerLogic main window, just above the Status bar.

To open a toolbar, use the Standard or Object Selector command under the Window menu. Choosing the command again will close the toolbar.

Standard Toolbar

Open Icon

Displays the File Open dialog box.

Save Icon

Saves the changes to the current schematic. If this is a new, unnamed schematic, the File Save As dialog box appears.

Sheet List Box

Displays the sheet set of the current schematic in a drop-down list box on the Standard toolbar. To change the sheet in the work area, choose a sheet from this list. Sheets that contain objects selected using the Object Selector display in bold.

Select Icon

Places PowerLogic in default Autoselect mode.

Copy Mode Icon

Invokes Copy mode.

Move Mode Icon

Places PowerLogic in Move mode.

Delete Mode Icon

Places PowerLogic in Delete mode.

Query Mode Icon

Places PowerLogic in Query/Modify mode.

Drafting

Displays the Drafting toolbox.

Design

Displays the Design toolbox.

Busses

Displays the Busses toolbox.

Undo

Rescinds the last editing action such as move, delete, swap, add connection, create 2D line, and others.

Redo

Repeats the last editing action made prior to an undo action.

Zoom

Choosing the Zoom icon invokes Zoom mode. The magnifying glass cursor indicates Zoom mode. To zoom in, locate the cursor to indicate the center of the zoom area and press the left button. To zoom out, locate the cursor and press the right button.

You can specify the zoom area. Press and hold the left button and drag diagonally upward to zoom in, diagonally downward to zoom out.

Sheet

Adjusts the display to fit the sheet outline in the work area. Information that is external to the sheet outline, such as dimensions and notes, does not display.

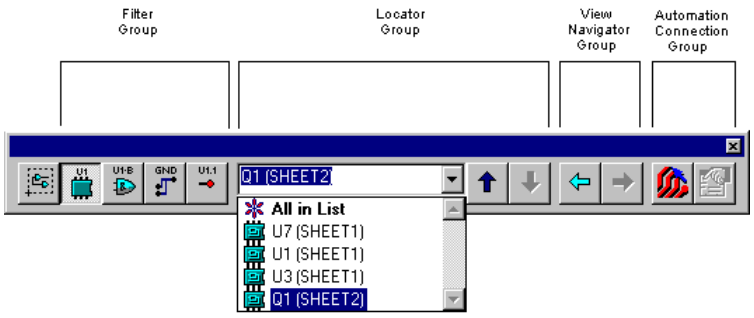
Redraw

Refreshes the work area display without modifying the current zoom factor or coordinate position.

Object Selector Toolbar

The Object Selector toolbar is visible, by default, at the bottom of the PowerLogic main window and is available only in Schematic Editor mode. Open or close the toolbar using the Object Selector command under the Window menu.

The Object Selector toolbar contains the following groups of tools:



Group Select Mode

Places PowerLogic in Group Select mode. See "Group Operations" in the "Design" chapter for more information.

Part Filter

The Filter group contains icons that allow you to select objects based on type. You can select an entire part (all gates), a single gate of a part, a net, or a pin. When you select an object on one sheet of the schematic, the object is selected across all sheets of the design. Sheets that contain objects selected using the Object Selector display in bold in the Sheet list box. Additional selection filters are available on the Selection pop-up menu.

To select an entire part in a schematic, choose the Part Filter icon, and then select any gate on the part.

Note: The Part Filter and the Gates Filter cannot be invoked at the same time.

Gates Filter

The Filter group contains icons that allow you to select objects based on type. You can select an entire part (all gates), a single gate of a part, a net, or a pin. When you select an object on one sheet of the schematic, the object is selected across all sheets of the design. Sheets that contain objects selected using the Object Selector display in bold in the Sheet list box. Additional selection filters are available on the Selection pop-up menu.

To select any single gate on a part, choose the Gates Filter icon, and then select the gate. To select multiple gates, choose the Gates Filter icon, and then use the Ctrl+click or drag-to-select methods to add gates to the selection.

Note: The Part Filter and the Gates Filter cannot be invoked at the same time.

Net Filter

The Filter group contains icons that allow you to select objects based on type. You can select an entire part (all gates), a single gate of a part, a net, or a pin. When you select an object on one sheet of the schematic, the object is selected across all sheets of the design. Sheets that contain objects selected using the Object Selector display in bold in the Sheet list box. Additional selection filters are available on the Selection pop-up menu.

To select a net, choose the Net Filter icon, then select any connection, pin, or net name on the net.

Pin Filter

The Filter group contains icons that allow you to select objects based on type. You can select an entire part (all gates), a single gate of a part, a net, or a pin. When you select an object on one sheet of the schematic, the object is selected across all sheets of the design. Sheets that contain objects selected using the Object Selector display in bold in the Sheet list box. Additional selection filters are available on the Selection pop-up menu.

To select a pin, choose the Pin Filter icon, then select the pin. To select multiple pins, choose the Pin Filter icon, then use the Ctrl+click or drag-to-select methods to add pins to the selection.

Object Locator List

The Object Locator group provides an efficient means of locating particular objects in your design.

The Object Locator drop-down list box displays all selected objects and the sheet on which they are located. When multiple objects are selected, the All in List command appears on the list. This feature makes it easier for you to locate a particular object, especially in a busy design. Just make a multiple selection in the general area of the object you want to locate, then select the particular object from the drop-down list box. Choose All in List to reselect all of the objects in the multiple selection.

Note: To locate a very small object, zoom in on the current sheet, then select the object from the drop-down list box.

Another way of locating an object is to type information about the object in the text area of the drop-down list box. Type the part name, gate name, part type name, net name, or pin name of the object you seek in the text area of the drop-down list box, then press Enter.

Next/Previous

The Object Locator group provides an efficient means of locating particular objects in your design.

Use the Next/Previous Object icons to move the selection from one object to the next in the current multiple selection.

Previous/Forward Sheet

These icons work like the navigation icons in an Internet browser. Choose the Previous Sheet icon to move to a previously viewed sheet. Choose the Forward Sheet icon to move to the next sheet in the design.

Connect

Makes an OLE connection to PowerPCB. Behaves exactly like choosing Tools/OLE PowerPCB Connection. See the "OLE PowerPCB Connection" topic in the "OLE in PowerLogic" chapter for more information. Once connected to PowerPCB, the Connection Properties icon becomes available.

Choosing this icon a second time disconnects from PowerPCB.

Connection Properties

Opens the OLE PowerPCB Connection dialog box. See the "OLE PowerPCB Connection" topic in the "OLE in PowerLogic" chapter for more information.

Toolboxes

Toolboxes are interchangeable toolbars that you can use to create and edit design items. You can only open one toolbox at a time.

To open a toolbox, use the Drafting Toolbox, Design Toolbox, or Bus Toolbox command under the Window menu. Choosing the command again will close the toolbox.

You can also open a toolbox by choosing the appropriate icon on the toolbar. Use these icons to open or close a toolbox.

Drafting — Used for operations on *drafting objects*, objects not generally associated with connectivity. See the "Drafting Operations" chapter for more information.

Design — Used for parts placement and connectivity operations. See the "Design Operations" chapter for more information.

Bus — Used for creating and modifying buses. See the "Design Operations" chapter for more information.

Another toolbox exists in PowerLogic, but you can only access it from the Part Editor:

Terminal — Used to add pins terminals to the decal and to assign pin names, pin number, pint types, and pin swapping information. See the "Library Operations" chapter for more information.

Dockable and Resizable Toolbar and Toolboxes

You can *dock*, or attach, the toolbars and toolboxes to any border of the main window. Additionally, you can leave these items *floating*, or unattached to a border, within the work area and move them around as needed for viewing purposes. You can size toolbars as well. For specific information on docking, moving, or resizing toolboxes and toolbars, refer to the Microsoft Windows Tutorial on-line. Search for toolbars.

ToolTips

ToolTips appear below icons and provide a command name or description for the icons in the toolbar and toolbox. To view a ToolTip, place and leave the cursor on an icon. ToolTips are inactive during system processing.

Work Area and Grid Settings

The maximum work area is a square 54 inches by 54 inches. The current grid settings show on the message line at the bottom of the work area, between the current default line width and the current cursor X and Y location. When you move an object or use a drafting command, the grid readout is replaced by a *Delta* X and Y reading, calculated from the cursor selection point when the command starts. Negative numbers indicate left and downward.

Display Grid

PowerLogic uses a dot grid as a drafting aid. You can set this field of white dots called the *display grid* to match your design grid, or set it at larger multiples of the design grid. Use the Global tab in the Preferences dialog box to set the display grid or use the GD modeless command. If you do not want to display the display grid, set it to 10.

Origin and Design Grid

When you start a new file, the default drawing format is centered in the work area with the origin, or 0,0 point, in the lower left corner. The origin appears as a large white dot. As you move the cursor, its position relative to the origin displays in the lower right corner of the screen. The numbers change in the multiples of the design grid. The minimum design grid increment is 2 mils.

The design grid intervals begin in all directions from the origin. To set the design grid, use the Design tab in the Preferences dialog box or use the G modeless command.

During design, if you move a part across the board, the cursor may move smoothly but the part *snaps* from grid point to grid point. When Snap to Grid is checked in the Status Window, you cannot place a part off of the current design grid.

To set the origin while in the Part Editor, choose Setup/Set Origin from the menu. Indicate the position of the new origin.

Controlling the View

You can use several methods to control which portion of the database is visible on the screen.

Using the View Commands and Scroll Bars

Use commands in the View menu to control the view.

- Use Zoom, or press the Zoom icon on the toolbar to enter Zoom mode.
  - To zoom in, position the cursor at the desired view center, and click the left mouse button.
  - To zoom out, position the cursor at the desired view center, and click the right mouse button.
  - To define a specific view area, click and hold the left mouse button, move the mouse to define the view rectangle, and release.
- Use Sheet, or press the Drawing Format icon on the toolbar, to redraw the current view.
- Use Redraw, or press the Redraw icon on the toolbar, to redraw the current view.
- Use Extents to fit all objects in the database on the screen.
- Use the scroll bars to cause the view to pan.

Using the Middle Mouse Button

Use the middle mouse button to perform panning and zooming operations:

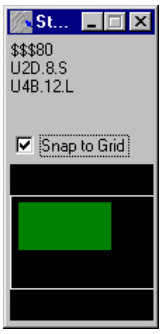
- To pan, click the middle mouse button. The view is centered at the cursor.
- To zoom in, click and hold the middle mouse button. Drag the cursor diagonally up. To adjust the bounding box, move the mouse. When you release the middle mouse button, the area within the bounding box displays.
- To zoom out, click and hold the middle mouse button. Drag the cursor diagonally down. This draws an inner box, representing the current view. Moving the cursor adjusts the outer box, which represents the new view. The relative size of the outer box to the inner box determines how far the view zooms out. Release the middle mouse button to complete the zoom out operation.
- To view the entire design, click and hold the middle mouse button, drag the cursor horizontally, and release.

Status Window

The status window is a floating window that contains:

- The postage stamp area. Use it to pan and zoom.
- Snap to Grid check box. Causes the cursor to snap to the current design grid if checked. For more details, see the "Setting Preferences" topic in the "Design Setup" chapter.
- The net name and pin connects of a selected connection and part details of a selected part.

To display the status window, choose Window/Status.



Using the Postage Stamp

The postage stamp at the bottom of the status window uses a color rectangular image to show the relationship between the drawing format and the current view. In the default startup state, you can use the postage stamp to relocate the current zoom view to another part of the design. To move the view, use the left mouse button to click the part of the postage stamp, which represents area you want to view. The work area pans to that section.

- The white rectangle represents the drawing format outline.
- The empty black rectangle represents the screen view.
- The postage stamp perimeter represents the viewing bit map. When you redraw, the graphics are actually stored in the viewing bit map; you can think of this as an invisible screen that is larger than the screen you can see. A part of the bit map is copied to the screen to view. When you pan, a different part of the bit map is copied to the screen; this is how smooth scrolling is implemented.

To pan the view:

1. Move the cursor over the drawing format on the postage stamp, and click and hold the left mouse button.
2. Move the cursor to drag over the outline. The view changes while you drag.
3. When you see the view you want to use, release the left mouse button.

To resize the view:

1. Position the cursor on the postage stamp at one corner of the new viewing rectangle.
2. Click and hold the right mouse button. Move the cursor to define the rectangle. Release the right mouse button.

Using the Numeric Keypad

You can also use the numeric keypad, located on the far right of most keyboards, to control the view. The Num Lock light can be on or off, except where noted.

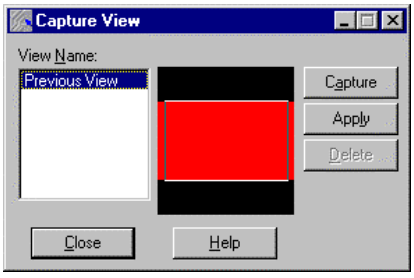
Press	To
Home	Fit the board to the view.
End	Redraw the current view.
Arrows	With Num Lock On, pan the viewing window. Moves one-half the screen width in the direction of the arrow. With Num Lock Off, move the cursor on grid unit.
5	With Num Lock On, draw the zoom rectangle.
Pg Up	Zoom In centered at cursor location.
Pg Dn	Zoom Out centered at cursor location.
Ins	Center the view at current cursor location, without zooming.

1. Position the cursor on the postage stamp at one corner of the new viewing rectangle.
2. Click and hold the right mouse button. Move the cursor to define the rectangle. Release the right mouse button.

Saving and Restoring Views

You can save a work area view for easy restoration.

1. Arrange the work area to the view that you want to capture.
2. Choose View/Capture from the menu. The Capture View dialog box appears.



3. In the Capture View dialog box choose Capture.
4. Type a name in the Capture a new view text box and choose OK.

You can create up to nine views. The view names appear at the bottom of the View menu.

To restore the view to one previously captured, select its name from the View Name list box and choose Apply. When you change the view, PowerLogic automatically saves the previous view. Choose Previous View in the View Name list box and choose Apply to restore this view.

Note: Capture is not available while in the Part Editor.

Modeless Commands

You can set or change some settings and functions at any time using a code letter for the command, entering the new value, and pressing Enter. This is called a *Modeless Command*.

Modeless Commands usually apply to values that you change frequently during design. Use the Modeless Command G, for example to change the grid setting. Type G, the new setting, and press Enter. Below is a summary of the Modeless Commands in PowerLogic:

To show this help topic at any time while PowerLogic is running, type **?** and press Enter.

Note: (X,Y) = coordinates; (S) = text; (n) = number.

Grid Settings

G<*n*>	Sets the design grid, for example G50.
GD<*n*>	Dot grid setting, for example GD100.

Design and Hierarchy

HI	Invokes Hierarchical Push.
HO	Invokes Hierarchical Pop.
Q	Invokes Quick Measure.
R<*width*>	Set minimum (Real) display width for paths.
SH<*sheet*>	Select sheet name or number.
W<*width*>	Set current line width.

Search Commands

S<*string*>	Search for named item (pin, part or net), for example SU1.
S<*n*><*n*>	Search absolute at <n> <n>, for example S1000 1000.
SR<*n*><*n*>	Search relative X and Y, for example SR-200 100.
SRX<*n*>	Search relative X, for example SRX300.
SRY<*n*>	Search relative Y, for example SRY400.
SX<*n*>	Absolute move to <n>, current Y, for example SX300.
SY<*n*>	Absolute move to <n> current X, for example SY400.

Angles

AA	Any angle.
AD	Diagonal angle.
AO	Orthogonal angle.

Drafting Objects

HC	Circle shape draw mode.
HH	Path shape draw mode.
HP	Polygon shape draw mode.
HR	Rectangle shape draw mode.

Mouse Click Substitutes

M	Activates pop-up for current mode (right mouse button).
Spacebar	Activates a left mouse button click (to add corners, select items, complete etc.) at the current cursor location.

Various

?	Shows this help topic.
BMW	Opens the Media Wizard dialog box. BMW creates a macro with associated files based on your last PowerLogic session or your current session. BMW records a session using the next.log file to simplify the process of reproducing a problematic PowerLogic session. This is also a command line option. See the Starting PowerLogic section of this topic. BMW is associated with another modeless command, BLT. See the BMW and BLT topic below for information on creating media.
BLT	Opens the Log Test dialog box. BLT finds and runs the BMW macros to test or replay the media. See the BMW and BLT topic below for information on playing back the PowerLogic session.
F<*name*>	View specified file (for example, **F demo.eco**).
I	Runs Database Integrity Test.

BMW and BLT

BMW creates media based on either the current PowerLogic session or the previous PowerLogic session using the next.log file automatic record macro. When you use BMW as a command line option it only creates media of the previous session; use the modeless command to create media of your current session. If you need to send files or other information to PADS Technical Support, you can run BMW and send the resulting files; BMW copies all the necessary files. To use the BMW modeless command:

1. Start PowerLogic.
2. Type BMW and press Enter. PowerLogic checks to see if the powerlogic.ini file has the log feature enabled. If it is not enabled, the message *Log file generation is not enabled properly in your powerlogic.ini file. Do you want it automatically enabled for this and following PowerLogic sessions?* appears.
3. Choose Yes to enable BMW and continue. If you choose No or Cancel, BMW is not enabled and you return to the PowerLogic work area. When you choose Yes, the Media Wizard dialog box appears.

4. Choose whether to Create Media from Current Session or to Create Media from Previous Session. This selection selects the correct next.log file.
5. Type your initials in the User Initials field to personalize the files. If you do not enter your initials, a default is used. zz is the default.
6. Choose whether to Delete all actions before last save. This deletes all entries in the .log file between the first Open and the last Save command used for each design file in use. Deleting the lines helps to prevent replaying the wrong data.
7. Choose OK to continue. BMW creates a directory for the files, \padspwr\logicfiles\<*month*><*day*><*initials*><*sequential_letter*>. The *initials* are the initials you enter in the dialog box and *sequential_letter* is a letter that BMW uses to sequence the directories. BMW uses the letter A first. If you created a directory earlier in the same day, BMW uses the next alphabetical letter.

   BMW next copies the next.ini file and the next.log file to the directory. They are renamed *directory_name*.ini and *directory_name*.log, respectively. If you chose to create media from the previous session, BMW copies and renames nextback.ini and nextbak.log.

   Once BMW copies the files, it renames and copies any files related to the .log file (such as .pcb, .asc, .eco, .cam, or .edp) to the directory. The naming format is *directory_name#*.pcb. The original file extension is used. The # represents an incremented number for multiple files.

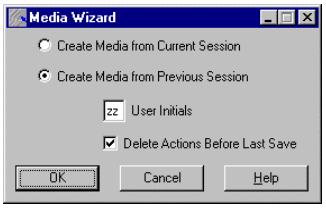
   When this process completes, the message *Media Wizard completed successfully* appears. You can run the Log Test to run the resulting BMW macro.

To run the BLT modeless command:

1. Type BLT and press Enter. The Log Test dialog box appears.
2. Choose a log test to run and choose OK.

   This copies the <directory_name>.ini file into the powerlogic.ini file and clears the PowerLogic work area. The <directory_name>.log file appears in the Macro dialog box.
3. Use the Macro dialog box to play back the .log file.

Media Wizard Dialog Box



Create Media from Current Session

Creates media, or prepares the related files, from the current PowerLogic session. This selection determines which next.log file is used.

Create Media from Previous Session

Creates media, or prepares the related files, from the previous PowerLogic session. This selection determines which next.log file is used.

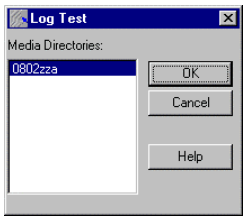
User Initials

Type your initials in this field to personalize the name of the files. Using your initials ensures that you don't confuse your media files with another user's files. If you do not enter your initials, the default is zz.

Delete Actions Before Last Save

Deletes all entries in the .log file between the first File/Open and the last File/Save command for each related design file. Turning this option on helps to prevent replaying the wrong data.

Log Test Dialog Box



Media Directories

Displays all of the media directories. A directory is created each time you use BMW. Choose a directory from the list and choose OK to open and run the files in that directory.

Shortcut Keys

For mouseless operation, use keyboard shortcuts to start commands for selected items and change some system settings. Following are the shortcut assignments for PowerLogic.

Control Key Shortcuts

Ctrl-B	View sheet
Ctrl-C	Copy
Ctrl-D	Redraw
Ctrl-E	Move selected group
Ctrl-F	Flip selected on X axis
Ctrl-N	New file

Ctrl-O	Open file
Ctrl-P	Print/Plot
Ctrl-Q	Query/Modify
Ctrl-R	Rotate selected
Ctrl-S	Save
Ctrl-V	Paste
Ctrl-W	View Zoom mode
Ctrl-X	Cut
Ctrl-Alt-C	Display Colors Setup
Ctrl-Alt-E	View Extents
Ctrl-Alt-G	Preferences
Ctrl-Alt-M	Toggle menu bar
Ctrl-Alt-P	View Previous
Ctrl-Alt-S	Status Window
Ctrl-Tab	Cycle Alternate Gate Decals
Ctrl-PageDown	Reset delta coordinates to measure from current position
Shift-Ctrl-D	Draw Group while in Move Group Mode
Shift-Ctrl-F	Flip selected on Y axis
Shift-Ctrl-P	Record prompt windows toggle (macros)

Function Key Shortcuts

F1	Help
F2	Add Connection
F8	Mouse move compression (macros)
F9	Absolute/Relative coordinates toggle (macros)
F10	End recording (macros)

Other Key Shortcuts

BackSpace	Remove last connection corner
Esc	Cancel command
M	Right mouse click
Spacebar	Left mouse click
Alt-Spacebar	Add Offpage while in Add Connection mode
Shift-Spacebar	Add Power Symbol while in Add Connection mode
Ctrl-Spacebar	Add Ground Symbol while in Add Connection mode

Pop-Up Menus

Pop-up menus provide an alternate method of choosing design, drafting, and object selection commands. Open a pop-up menu by pressing the right mouse button over the work area. The commands that appear on a pop-up menu depend on the object, or objects selected.

- Part or gate selected
- Net selected
- Pin selected
- Nets and parts selected
- No objects selected
- A Group of objects selected

Note: Object selection also effects which Verb commands are available on the Standard toolbar.

Pop-Up Menu Commands

Add Item

When you choose the Add Item command from the pop-up menu, a cascading menu with the following commands appears.

Add Part — Adds a part to the schematic. Similar to choosing the Add Part icon from the Design toolbox. See the "Design Operations" chapter for more information.

Add New Hierarchy — Adds a new hierarchical symbol to the schematic. Similar to choosing the New Hierarchical Symbol icon from the Design toolbox. See the "Design Operations" chapter for more information.

Add Connection — Adds a connection to the schematic. Similar to choosing the Add Connection icon from the Design toolbox. See the "Design Operations" chapter for more information.

Add Bus — Adds a bus to the schematic. Similar to choosing the Add Bus icon from the Bus toolbox. See the "Design Operations" chapter for more information.

Add Text — Adds a free text item to the schematic. Similar to choosing the Add Text icon from the Drafting toolbox. See the "Drafting Operations" chapter for more information.

Add Line Item — Adds a 2D line item, such as a polygon, circle, rectangle, or path. Also, specifies the drafting angle for, and adds arcs to, polygons or paths. Similar to choosing the Create 2D Line icon from the Drafting toolbox. See the "Drafting Operations" chapter for more information.

Attribute

Opens the Query/Modify Part Attributes dialog box. See the "Query/Modify Part" topic in the "Design Operations" chapter for information.

Cancel

Closes the pop-up menu without choosing a command.

Copy

Places PowerLogic in Copy mode. Not available when a single part, or multiple gates, are selected.

Copy to File

Saves the group to a file for use in another design. See "Group Operations" in the "Design" chapter for more information.

Cut Conns at Edge

Includes or excludes connections that extend beyond the group outline.

Delete

Deletes the selected parts, gates, or group. See "Group Operations" in the "Design" chapter for more information on deleting groups.

Edit Part

Opens the PowerLogic Part Editor. Available when a single part, or multiple gates from a single part, are selected. Not available when multiple parts, or multiple gates from more than one part, are selected.

Group Copy

Copies the group to a new location. See "Group Operations" in the "Design" chapter for more information.

Horiz Justify

Moves the insertion point across the text string for left, right, or center justification.

Modify Bus

When you choose the Modify Bus command from the pop-up menu, a cascading menu with the following commands appears.

Split Bus	Bisects a selected segment with a new, perpendicular segment.
Extend Bus	Continues the bus definition from either end point.
Move Bus Segment	Repositions the selected bus segment.
Delete Bus Segment	Removes the selected bus segment.

See the "Bus Toolbox" topic in the "Design Operations" chapter for more information on these commands.

Modify Line Item

When you choose the Modify Line Item command from the pop-up menu, a cascading menu with the following options appears.

Modify Line Item	Modifies line items by: pulling an arc from a segment or corner, splitting a segment, changing the line width of an item, filling a closed polygon, or changing the line style to solid or dotted.
Combine/Uncombine	Combines selected line items into a complex line item, or breaks combined line items into individual items.
Add from Library	Adds a 2D line item from a library.
Save to Library	Save a 2D line item to a library.

See the "Drafting Toolbox" topic in the "Drafting Operations" chapter for more information on these commands.

Modify Part

When you choose the Modify Part command from the pop-up menu, a cascading menu with the following commands appears.

Edit Part/Hierarchy	Select this command, and then select a part in the schematic to open the Part Editor. Starts the Part Editor, disabling the PowerLogic Schematic Editor
Swap Reference Designator	With no objects selected, choose Modify Part/Swap Ref-Des from the pop-up menu, and then choose the parts you want to swap.

Swap Pins	This command appears on two pop-up menus. With one pin selected using the Pins Filter on the Object Selector toolbar, choose Swap Pins from the pop-up menu. With no objects selected, choose Modify Part/Swap Pins from the pop-up menu, and then choose the two pins you want to swap. A prompt provides you with the opportunity to keep the connections tied to the current electrical pin, or to swap the electrical connections along with the pin number.

Move

Places PowerLogic in Move mode. Not available when a single part, or multiple gates, are selected.

When a group is selected, this pop-up menu command moves the group to a new location. See "Group Operations" in the "Design" chapter for more information.

Query/Modify

Places PowerLogic in Query/Modify mode. Not available when multiple parts or multiple gates from more than one part are selected.

Rotate 90

Changes the orientation of the text in 90-degree increments.

Select All

Selects all objects on all sheets using the current filter. For example, if the Parts Filter is turned on, choose Select All to select all of the parts in the entire schematic design.

Select Anything

Turns on the Gates, Nets, and Pins Filters. Turns off the Part Filter.

Select Gates

Selects the gates on the currently selected net; deselects the net.

Select Gates

Turns on the Object Selector Gates Filter. See the "Object Selector Toolbar" topic in this chapter.

Select Group

Turns off all Object Selector Filters and places PowerLogic in Group select mode. See the "Object Selector Toolbar" topic in this chapter.

Select Net

Selects the net to which the pin is connected.

Select Nets

Turns on the Object Selector Net Filter. See the "Object Selector Toolbar" topic in this chapter.

When a part or gate is selected, this pop-up menu command selects the net to which the currently selected part or gate is connected.

Select Part

Selects the entire part to which the currently selected gate or pin belongs.

Select Parts

This command performs two different functions, depending on what type of object is selected:

- Selects only the parts out of all of the currently selected objects.
- Turns on the Object Selector Parts Filter. See the "Object Selector Toolbar" topic in this chapter.

Select Pins

Turns on the Object Selector Pins Filter. See the "Object Selector Toolbar" topic in this chapter.

Set Origin

Changes the insertion point of the group.

Set Verb Mode

You can only set the verb mode when no objects are selected. When you choose the Set Verb Mode command a cascading menu with the following commands appears.

Select Mode	Places PowerLogic in Group Select mode.
Copy Mode	Places PowerLogic in Copy mode.
Move Mode	Places PowerLogic in Move mode.
Delete Mode	Places PowerLogic in Delete mode.
Query Mode	Places PowerLogic in Query mode.

See the "Modes of Operation" topic in the "Design Operations" chapter for more information on these commands.

Show Attr Name

Displays the name of the attribute as well as the value as NAME=VALUE. Use this option to set unique display of attributes in a design. A check appears next to the option when turned on.

See the "Query/Modify Part Text Visibility" topic in the "Working With Attributes" chapter for information on setting display properties for all attributes in the design.

Show Rules

Opens the Net Rules dialog box, where the Nets list box displays the currently selected net(s). See the "Net Rules" topic in the "Design Rules and Layers" chapter for more information.

Size

Opens the Text Size dialog box where you can change the height and width of the text string.

Swap Pins

This command appears on two pop-up menus:

- With one pin selected using the Pins Filter on the Object Selector toolbar, choose Swap Pins from the pop-up menu.
- With no objects selected, choose Modify Part/Swap Pins from the pop-up menu, and then choose the two pins you want to swap.

A prompt provides you with the opportunity to keep the connections tied to the current electrical pin, or to swap the electrical connections along with the pin number.

Vertical Justify

Moves the insertion point up the text string characters for top, center, or bottom of character placement.

Visibility

Opens the Part Text Visibility dialog box. See the "Query/Modify Part Text Visibility" topic in the "Working With Attributes" chapter for information.

LICENSE AGREEMENT AND LIMITED PRODUCT WARRANTY

PADS Software, Inc. ("PADS") and the undersigned customers (The "Customer") hereby agree that the Software Products being licensed to the Customer, and as amended from time to time, are the property of and embody proprietary technology owned by or licensed to PADS. Unauthorized uses and copying of the Licensed Products are prohibited by law, including United States and foreign copyright law. The price paid by the Customer for the Licensed Products constitutes a license fee that entitles the Customer to use the Licensed Products as set forth below.

1. LICENSE COVERAGE:

PADS hereby grants the Customer a non-exclusive right to use the Software Products provided by PADS at the Designated Location whether on a single computer or on a network, within the constraints of the original network license configuration. The Customer may physically transfer the licensed Products from one computer to another at the same Designated Location. The addition of any networked computers beyond that specified without notice and payment to PADS, shall breach this agreement. The rights granted to the Customer do not include the right to copy Licensed Products, except to create a backup copy, and do not include any right to access, use or modify the source code of the licensed Products for any purpose.

2. TITLE AND OWNERSHIP:

All intellectual property and proprietary rights for the licensed Products and related documentation shall at all times remain the sole and exclusive property of PADS and/or the relevant third parties.

3. TRANSFERABILITY:

The rights granted to the Customer under this Agreement are non-transferable without the prior written consent of PADS. Any attempted assignment or other transfer shall be null and void and PADS reserves the right to seek legal action against such attempts.

4. LICENSE TERM:

The license granted under this Agreement shall commence upon delivery of the licensed Products by PADS to the Customer. PADS may terminate this license at any time if the Customer neglects or fails (i) to pay all applicable license fees and maintenance fees within specified payment terms and (ii) to strictly adhere to any of the Customer's obligations under this Agreement. If this license is terminated by PADS, the Customer will, within 10 days of request by PADS, deliver to PADS either all product components including originals, updates and copies, related documentation, security devices (if applicable) and other materials, or an affidavit signed by an authorized representative of the Customer attesting to the fact that such have been completely destroyed.

5. INSTALLATION:

Licensee shall be solely responsible to install the Licensed Products unless otherwise agreed upon in writing.

6. PROHIBITED ACTS:

The Customer shall not, without the express prior written approval of PADS: (i) copy the Licensed Products except to load the Products into the computer or to make a backup copy for security purposes only; (ii) employ any device or instructions with respect to the licensed Products; (iii) modify, decode, reverse compile or disassemble the Licensed Products; (iv) transfer or assign or attempt to transfer or assign the licensed Products, related documentation or the rights under this Agreement except with the prior written consent of PADS; or (v) export or re-export the licensed Products or related documentation without all appropriate United States and foreign government licenses and approvals.

7. LIMITED WARRANTY:

7.1 Software: If PADS confirms a defect reported by Licensee in the unaltered Software, PADS shall use commercially reasonable efforts to provide a remedy. PADS does not warrant that the operation of any of the Licensed Products will be uninterrupted or error free, nor does it guarantee that its remedial efforts will correct a found defect.

7.2 Media and Peripherals: PADS warrants to the original licensees of each of its software Products that the media on which the program is recorded shall be free from defects in materials or manufacture under normal use for the standard period of (30) days from delivery. Products found to have a defect in their media will be replaced by PADS at no charge to the customer if returned to either the dealer from which such program was obtained or to PADS at its principal office (PADS Software, Inc., 165 Forest Street, Marlborough, MA 01752 U.S.A.). Returns must be postage prepaid and be accompanied by proof of the date of delivery. If the defect results from accident, abuse or mishandling, however, PADS will have no responsibility to replace the media.

EXCEPT AS EXPRESSLY PROVIDED ABOVE, ALL PRODUCTS AND RELATED DOCUMENTATION ARE PROVIDE BY PADS "AS IS" AND NO WARRANTY OF ANY KIND, INCLUDING BUT NOT LIMITED TO ANY WARRANTY OF MERCHANTABILITY OR FITNESS FOR A PARTICULAR PURPOSE, IS MADE OR AUTHORIZED TO BE MADE BY PADS.

THE FOREGOING WARRANTY GIVES YOU SPECIFIC LEGAL RIGHTS AND RIGHTS MAY VARY IN STATES OF UNITED STATES AND FOREIGN COUNTRIES. THE FOREGOING MAY LIMIT OR EXCLUDE THE RIGHTS THAT THE CUSTOMER WOULD OTHERWISE BE ENTITLED TO UNDER THE LAWS OF SOME STATES OR COUNTRIES. OTHER STATES OR COUNTRIES DO NOT ALLOW THE LIMITATION OR EXCLUSION OF LIABILITY FOR INCIDENTAL OR CONSEQUENTIAL DAMAGES SO THE ABOVE LIMITATION OR EXCLUSION MAY NOT APPLY.

8. SOFTWARE MAINTENANCE:

PADS shall provide, and Licensee may accept, software maintenance services under the Software Maintenance Agreement at PADS' then current maintenance fee for an initial term of one calendar year after expiration of the Software Warranty period, renewable year to year until termination occurs under the said Software Maintenance Agreement.

8.1 Annual Service Fee. Licensee shall pay PADS an annual service fee for each of the Licensed Products covered by this Agreement at the start of any applicable term.

8.2 Payments. All payments under this Agreement shall be due according to specified payment terms.

8.3 Renewals. Payment for renewals must be received by PADS at its principal offices prior to the applicable Anniversary Date. Non-payment by the Anniversary Date will constitute discontinuance of Maintenance and Support on such licensed Products. Licensee shall incur reinstatement charges in the event of a lapse and subsequent renewal.

8.4 Increases. The applicable Maintenance fee for each licensed program is subject to change from time to time. Such increase for annual service fees would occur at the beginning of any renewal term.

8.5 Termination. Termination of the Maintenance Agreement will end any support obligation of PADS.

8.6 Added Coverage Fees. If any of the Licensed Products are added for Maintenance under this Agreement after its term has commenced, PADS will pro rate the additional fee from the expiration of the System's 30-day warranty period through the remainder of the Agreement term.

8.7 Upgrade and Update Fees. Upgrades and Updates may require additional service fees. Upon receipt of any Upgrade or Update from PADS requiring such fees, Licensee shall pay an additional service fee equal to the amount by which the annual service fee for the Upgrade or Update exceeds the annual service fee for the Software, prorated for the remainder of the term of the agreement.

8.8 Add on products: Add-on products and modules for which Maintenance has not been purchased, will not be automatically entitled to Maintenance and future Updates.

8.9 Acceptance. Customer agrees to pay Maintenance on the full configuration purchased if Maintenance is accepted.

9. MAINTENANCE SERVICES:

9.1 Services Provided. The PADS Customer Support Department shall provide Licensee with the following services during the term or any renewal term of this Agreement for each of the Licensed Products covered under this Agreement, if Maintenance has been accepted and payments are in good standing:

9.1.1. Telephone diagnostics and assistance during PADS' normal business hours excluding weekends and holidays observed by PADS

9.1.2 Repair or replacement of media that PADS determines to be defective and eligible if returned to PADS in accordance with return policy

9.1.3 Delivery of new Updates upon their release

9.2 Timing of responses. PADS will use its commercially reasonable efforts to respond promptly to any reasonable service request by the Licensee, but shall determine solely the timing of its Update deliverables.

9.3 Additional services. PADS may at its discretion and Licensee's request, provide maintenance otherwise excluded under the Agreement on a per-call basis during normal business hours, excluding holidays observed by PADS, for which services Licensee shall be charged PADS' then current standard per-call rates.

10. EXCLUDED SERVICES:

The following Maintenance services shall not be provided under this Agreement but may be agreed upon at PADS' discretion.

10.1. Repairs to the Licensed Products necessitated by (a) relocation, movement, improper operation, neglect or misuse of the Licensed Products, (b) Licensee's failure to maintain proper site or environmental condition, (c) any fault of Licensee's agents or employees, (d) any attempts at repairs, maintenance, or modifications to the Licensed Products performed by other than authorized PADS service personnel, (e) casualty, natural or unnatural disasters, theft, loss, strikes, riot, war, the unauthorized acts of third parties, (f) failure or interruption of any electrical power, telephone or communication line or like cause, or (g) any other cause external to the Licensed Products except ordinary use.

10.2. Service at Licensee's location

11. INDEMNIFICATION/LIMITATION ON LIABILITY:

The Customer agrees to indemnify, defend and hold PADS harmless against any and all claims, costs, losses, damages, liabilities and expenses (including reasonable attorneys' fees and costs and reasonable amounts paid in settlement) incurred by PADS that arise out of or relate to any claim, suit, action or proceeding threatened or commenced by the Customer or any third party in respect of death or injury, damage or loss resulting from or which is alleged to result from the operation of the licensed Products or the system on which such are used, except to the extent it is determined that such claims, costs, losses, damages, liabilities and expenses resulted from the willful misconduct of PADS. IN NO EVENT WILL PADS BE LIABLE FOR ANY DAMAGES INCLUDING LOST PROFITS FOR ANY SPECIAL, INDIRECT, INCIDENTAL or CONSEQUENTIAL DAMAGES ARISING OUT OF THE USE OR INABILITY TO USE THE PRODUCT, WHETHER CLAIMED UNDER THE LICENSE AGREEMENT OR OTHERWISE.

12. ACKNOWLEDGMENT:

The Customer acknowledges that computer aided design is a complex undertaking requiring coordination between many different hardware and software elements, including elements not supplied by PADS, and requiring proper use and caution on the part of operators and others.

13. MISCELLANEOUS:

This Agreement shall be governed by the Laws of the Commonwealth of Massachusetts. In the event of any dispute or claim arising out of or with respect to this Agreement, the prevailing party shall also be entitled to recover its reasonable attorneys' fees and costs. It is understood and agreed that this Agreement contains the entire and only understanding and agreements between the parties relating to the subject matter of this Agreement, promise or condition not contained in this Agreement shall not be binding on either party. The terms of this agreement shall not be modified except in writing and signed by each party.

Copyright © 1997-1999 PADS Software, Inc. All rights reserved.
165 Forest Street
Marlborough, MA 01752
U.S.A.

PowerLogic
User's Guide

PADS Software, Inc. has taken due care in preparing this documentation. Due to lead times required for printing, some minor functionality differences may exist. Check your release notes and on-line help for updated information.

PADS Software, Inc. and the program authors have no liability to the purchaser or any other entity, with respect to any liability, loss, or damage caused, directly or indirectly by this software, including but not limited to, any interruptions of service, loss of business, anticipatory profits, or consequential damages resulting from the use of or operation of this software.

Updates may be made to this documentation and incorporated into later editions.

ActiveX™ is a trademark of the Microsoft Corporation.
Adobe® Acrobat® Reader is a registered trademark of Adobe Systems, Inc.
C-LINK® is a registered trademark of CAE Technologies, Inc.
DXF® and AutoCAD® are registered trademarks of Autodesk, Inc.
FLEX*lm*® is a registered trademark of Globetrotter Software, Inc.
InstallShield® is a registered trademark of the InstallShield Corporation.
Microsoft® and MS-DOS® are registered trademarks of the Microsoft Corporation.
PostScript™ is a trademark of Adobe Systems, Inc.
Pro/ENGINEER® is a registered trademark of Parametric Technologies Corporation.
Sax Basic Engine™ is a trademark of Sax Software Corporation.
SPECCTRA® is a registered trademark of Cadence Design Systems, Inc.
unInstallShield™ is a trademark of the InstallShield Corporation.
ViewDraw® and Workview® are registered trademarks of Viewlogic Systems.
VBScript™, Visual Basic®, Visual C++®, Windows™, and Windows NT® are trademarks of the Microsoft Corporation.

All products mentioned herein are trademarks of their respective companies.

First Printing April 1997
Second Edition June 1999
Printed in the United States of America

Contents

Preface

Welcome to PowerLogic 3.0 xvi
Conventions xvi
Design Conventions xvi
Documentation Conventions xvii
Frequently-Asked Questions xxi
Memory and Hardware xxi
Installation Questions xxi
Compatibility with Older Design and Support Files xxi
Questions on General PowerLogic Operations xxii

Chapter 1
The PowerLogic Interface

Starting PowerLogic 1-1
Start-up Options 1-1
Using the Interface 1-3
Menu Commands 1-4
File Menu 1-4
Edit Menu 1-5
View Menu 1-8
Setup Menu 1-9
Tools Menu 1-10
Window Menu 1-11
Help Menu 1-12
Toolbars and Toolboxes 1-12
Toolbars 1-12
Toolboxes 1-17
Dockable and Resizable Toolbar and Toolboxes 1-17
ToolTips 1-17

Work Area and Grid Settings....1-18
Display Grid....1-18
Origin and Design Grid....1-18
Controlling the View....1-18
Using the View Commands and Scroll Bars....1-19
Using the Middle Mouse Button....1-19
Status Window....1-19
Saving and Restoring Views....1-21
Modeless Commands....1-22
BMW and BLT....1-23
Media Wizard Dialog Box....1-24
Log Test Dialog Box....1-25
Shortcut Keys....1-25
Control Key Shortcuts....1-25
Function Key Shortcuts....1-26
Other Key Shortcuts....1-26
Pop-Up Menus....1-27
Pop-Up Menu Commands....1-27

Chapter 2
File Operations

Opening Files....2-1
File Open Dialog Box....2-1
New Files....2-1
Saving Files....2-2
Importing and Exporting Files....2-2
Importing Files....2-2
Exporting Files....2-3
OLE Import/Export....2-3
ASCII Output Options....2-4
ASCII File Format....2-5
Creating a Netlist....2-5
Netlist to PCB Dialog Box....2-6
ECO Operations....2-8
ECO Forward and Backward Annotation....2-8
Forward ECO to PCB....2-8
Backward Annotate from PCB....2-12

Part Level Backward Annotation 2-13
Gate Level Backward Annotation 2-14
Pin Level Backward Annotation 2-14
Net Level Backward Annotation 2-15
Attribute Level Backward Annotation 2-15
ECO File Format 2-16

Chapter 3
Design Setup

Setting Preferences 3-1
Global Tab 3-2
Design Tab 3-4
Heights/Widths Tab 3-6
Setting Colors 3-7
Customizing PowerLogic 3-9
default.txt 3-9
width.dat 3-9
future.dat 3-10
The Macro Recorder 3-10
Macros and Dialog Boxes 3-11
Macro Files 3-11
Macro Dialog Box 3-12
Creating a New Macro 3-14
Assigning Shortcut Keys 3-16
Macro Playback 3-17
Recording a PowerLogic Session 3-18

Chapter 4
Design Operations

Starting a New Design 4-1
Design Toolbox 4-1
Adding Parts 4-2
Adding Connections 4-5
Hierarchical Design 4-8
Creating a Hierarchical Symbol 4-9
Creating a Top-down Hierarchy 4-10
Creating a Bottom-Up Hierarchy 4-11

Modifying a Hierarchical Symbol....4-12
Swapping Reference Designators....4-12
Swapping Pins....4-13
Bus Toolbox....4-13
Adding a Bus....4-14
Splitting a Bus....4-15
Extending a Bus....4-15
Moving a Bus Segment....4-15
Deleting a Bus Segment....4-15
Sheets....4-16
Modes of Operation....4-17
Group Select Mode....4-17
Verb Mode....4-17
Object Select Mode....4-21
Group Operations....4-22
Setting a New Origin for the Group....4-22
Removing Connections Outside the Boundary....4-22
Adding to or Removing from a Group....4-22
Moving a Group....4-23
Copying a Group....4-23
Displaying Groups During Move or Copy....4-24
Deleting a Group....4-24
Copying a Group to a File....4-25
Pasting a Group from a File....4-25
Preserving Reference Designators....4-25
Cut, Copy, and Paste....4-26
Copy as Bitmap....4-26
Query/Modify....4-26
Query/Modify Part....4-27
Change Part Type Dialog Box....4-30
Query/Modify PCB Decal Assignment....4-31
Get Decal from Library Dialog Box....4-32
Query/Modify Net Name....4-33
Query/Modify Bus Name....4-34
Query Hierarchical Component....4-34
Editing Off-Page Symbol Sheet Numbers Per Line....4-35

Chapter 5
Working With Attributes

Attributes 5-1
Creating New Attributes in a Schematic 5-2
Manage Schematic Attributes Dialog Box 5-2
Add New Attributes Dialog Box 5-4
Managing Attributes in a Schematic 5-4
Query/Modify Part Attributes 5-5
Query/Modify Part Text Visibility 5-7
Using Attribute Labels 5-8
Creating Attribute Labels 5-9
Add Attribute Label Dialog Box 5-9

Chapter 6
Library Operations

Library Manager 6-1
Selecting a Library 6-2
Library Data Types 6-2
General Purpose Library Operations 6-2
Edit Operations 6-3
Search Filter 6-4
Modifying the Library List Order 6-4
Managing Attributes in the Library 6-6
Creating Attributes in the Library 6-6
Manage Library Attributes Dialog Box 6-7
Add New Attribute to Library Dialog Box 6-9
Browse Library Attributes Dialog Box 6-10

Chapter 7
Part Editor Operations

The Part Editor 7-1
Changing and Updating Library Parts 7-2
Creating Single Gate Parts 7-2
Constructing the Decal 7-3
Assigning Electrical Information to the Part 7-4
Saving the Part to the Library 7-4

Creating Multigate Parts 7-7
Assigning Pin Information to the CAE Decal 7-8
Select Gate Decal 7-8
Creating a New CAE Decal 7-9
Part Information for Part 7-10
General Tab 7-11
Editing Logic Families 7-13
PCB Decals Tab 7-14
Gates Tab 7-16
Assign Decal to Gate Dialog Box 7-19
Signal Pins Tab 7-21
Attributes Tab 7-22
Alphanumeric Pins Tab 7-24
Connector Tab 7-25
Creating New Parts from Existing Parts 7-26
Select Type of Editing Item 7-27
Creating a New Connector 7-28
Assign Alternatives for Connector 7-29
Browse for Special Symbols 7-29
Creating a New Pin Decal 7-30
Managing Pin Decals 7-31
Get Gate Decal from Library 7-32
Get Part Type from Library 7-33
Part Editor Drafting Toolbox 7-34
Decal Wizard 7-34
Terminal Toolbox 7-35
Adding Terminals 7-36
Changing Pin Decals 7-37
Setting Pin Numbers 7-37
Changing Pin Numbers 7-37
Setting Pin Names 7-38
Changing Pin Names 7-38
Setting Pin Types 7-38
Setting Pin Swaps 7-39
Changing Sequence Numbers 7-39

Special Symbols....7-40
Creating New Special Symbols....7-40
Select Pin Decal....7-41
Assign Alternatives for Off-page Part....7-41
Assign Alternatives for Power Part....7-42
Assign Alternatives for Ground Part....7-43
Creating New Special Symbols from Existing Symbols....7-44
Special Symbol Naming Conventions....7-45
Updating Off-page Symbols from the Library....7-45
Updating Pins from Library....7-45
Save Off-page Symbols to Library....7-46
Managing Special Symbols in the Library....7-46
Setting the Origin for a Part....7-46
Check Part....7-46

Chapter 8
Drafting Operations

Drafting Toolbox....8-1
Adding Text....8-2
Creating 2D Line Items....8-4
Modifying 2D Line Items....8-5
Combining 2D Line Items....8-7
Adding Drafting Items from a Library....8-7
Adding Drafting Items to a Library....8-8
Query/Modify....8-8
Query/Modify Text....8-8
Query/Modify Drafting Object....8-9

Chapter 9
OLE in PowerLogic

OLE Functions in PowerLogic....9-1
OLE Terminology....9-2
Embedded Objects....9-2
Linked Objects....9-2
OLE Automation....9-2
Visual Editing....9-2
Sizing handles....9-2

OLE, Object Linking and Embedding 9-3
Insert Object Dialog Box: Create New 9-3
Insert Object Dialog Box: Create from File 9-4
Displaying OLE Objects 9-6
OLE and View Menu Commands 9-6
Turning the Display of OLE Objects Off 9-6
Redrawing a Screen Containing OLE Objects 9-7
Changing the OLE Object Background Color 9-7
Selecting OLE Objects 9-7
Moving and Sizing OLE Objects 9-8
Cutting, Copying, and Pasting OLE Objects 9-8
Editing OLE Links 9-8
OLE Visual Editing 9-10
Converting OLE Objects and other OLE-specific Editing Commands 9-10
Edit 9-11
Open 9-11
Convert 9-11
Saving OLE Objects 9-12
Deleting OLE Objects 9-13
OLE and Print/Plot 9-13
OLE PowerPCB Connection 9-13
OLE Receive Tab 9-14
OLE Send Tab 9-14
OLE Design Tab 9-15
OLE Document Tab 9-16
OLE Preferences Tab 9-16
Visual Basic Scripting 9-17
Sax Basic Engine Dialog Box 9-18
Visual Basic Scripts Dialog Box 9-21
Visual Basic Sample Scripts 9-22

Chapter 10
Design Rules and Layers

Setting Design Rules 10-1
Rules Hierarchy 10-2
Default Rules 10-3
Class Rules 10-4
Net Rules 10-6

Conditional Rules 10-7
Differential Pairs 10-9
Rules Report 10-10
Clearance Rules 10-11
Routing Rules 10-12
High-speed Rules 10-15
Export Rules to PCB 10-18
Import Rules from PCB 10-18
Setting Layers 10-19
Layer Thickness Dialog Box 10-20
Plane Layer Net Assignments 10-21
Increasing the Number of Electrical Layers 10-22
Reassigning Electrical Layer Data 10-22

Chapter 11
Printing and Plotting

Print/Plot Operations 11-1
Select Items 11-3
Plot Options 11-5
Pen Plotter Setup 11-6
Pen Plotter Advanced Setup 11-7
Photo Plotter Setup 11-8
Photo Plotter Advanced Setup 11-9
PostScript™ Printing to a File 11-11
Creating Reports 11-11
Unused 11-13
Part Statistics 11-13
Net Statistics 11-14
Job Limits 11-14
Off Page 11-16
Bill of Materials 11-17

Index In-1

Preface

Welcome to PowerLogic 3.0

PADS-PowerLogic is the next generation of schematic capture from PADS Software, Inc. PADS developed the graphical user interface according to Microsoft® Windows™ standards. From the flat (one level) menu structure to the use of standard shortcut keys and a context-sensitive On-line Help system, PowerLogic's design maximizes ease of use.

Conventions

The following is a list of design and documentation conventions to help familiarize you with PowerLogic and the documentation.

Design Conventions

For commands that operate on a selected object, you must place PowerLogic in Verb mode before selecting an object. Group operations are the only commands that let you choose the objects first. PowerLogic is in Verb mode whenever you choose one of the toolbar icons. For example, to move a part, choose the Move Mode icon, and then select the object to move.

Units of measure are in mils, unless stated otherwise.

The minimum PowerLogic database unit is 2 mils. The maximum size of a schematic is 54 by 54 inches.

Documentation Conventions

Typographical Guidelines

Bold	Indicates characters that you must type exactly as they appear.
Italic	Indicates messages from PowerLogic, variables that you must specify, new terms, and book titles.
`Courier`	Indicates output from PowerLogic or the contents of a file.

Mouse and Keyboard Actions

Double-click	To position the cursor on an item and rapidly press and release the left mouse button twice.
Select	To highlight or target a design or text object for the next command execution. Position the cursor on a design or text object and click the left mouse button. Selecting alone does not start a command or action (refer to Verb mode).
Indicate	To specify a point of a design object during its creation or modification. Position the cursor at the appropriate location and click the left mouse button. Move the cursor and repeat as necessary.
Choose	To pick a menu command, icon or dialog box option, button, or tab to carry out an action. Position the cursor on the menu item or dialog box item and click the left mouse button.
Drag	To locate a design or text object, or to specify a zoom area. Position the cursor at the starting point or on the object, press and hold the left mouse button while you move the cursor, and release the mouse button.
Type	To type characters. Type in a text box to provide names and values.
Enter	To type characters and press Enter usually at a prompt (usually not in a dialog box, because pressing Enter executes and closes the dialog box).
Press	To press keys on the keyboard or, if specifically stated, keys on the keypad.

GUI Elements/Windows Conventions

Menu bar	A command list across the top of the application window that changes depending on whether the PowerLogic Schematic Editor or Part Editor is active.
Pop-up menus	A command list that appears at the cursor location when you click the right mouse button. Pop-up menus can also contain cascading menus.
Command button	A labeled button that represents a command or command sequence and when chosen immediately invokes the command(s).
Text box	A dialog box area where you type text to supply variable information.
List box	A dialog box area containing a list from which, based on context sensitivity, you can choose a single item or multiple items. Scrolling enables you to view the entire list if it is too large for the display.
Drop-down list box	A dialog box area containing a single item list box that requires you to select the down arrow to view the hidden contents.
Multicolumn list box	A dialog box area containing a tabular list area that contains multiple items with multiple properties. You can individually choose each item and property.
Check boxes	A dialog box option that is individually on (selected) or off. A check mark in the box indicates the selected option(s).
Option buttons	A group of mutually exclusive dialog box options. A filled button indicates the selected option.
Toolbar	An icon list across the top of the application window that changes depending on whether the PowerLogic Schematic Editor or Part Editor is active.
Toolbox	A secondary icon list beneath the toolbar that appears, changes, or disappears depending on which icon you choose from the toolbar.
Grayed Text	Indicates, based on context sensitivity, unavailable options or commands.

General Commands

The following commands appear in several PowerLogic dialog boxes. The functionality of these commands is identical in each occurrence:

OK	Accepts and executes the changes and closes the dialog box.
Apply	Accepts and executes the changes without closing the dialog box.
Cancel	Cancels all changes and closes the dialog box.
Help	Opens a help window and displays help text for the current operation. Press F1 if no Help button exists.

The Filter area occurs throughout PowerLogic dialog boxes. The functionality of the Filter is similar, if not identical, in each occurrence. Filter options include:

Library	Specifies library directories to search for.
Items	Specifies the prefix, specific part, or item name for which to search libraries. Libraries are searched for names that match the wildcard text string you type. Type the prefix that you want to search for, followed by an asterisk; for example, type DIP* to view all parts that begin with DIP. Type * to view all parts or items in the specified libraries.
Apply	Searches the libraries and displays the search results.

Frequently-Asked Questions

Here are answers to frequently asked questions that many new users have about PowerLogic installation and usage.

Memory and Hardware

What are PowerLogic's hardware and operating system requirements?

PowerLogic can run under Microsoft Windows 95 and Windows 98 with at least 16 MB of memory, or Windows NT with 32 MB of memory. PowerLogic requires about 30 MB of disk space for either operating system.

Does PowerLogic require a security device to run the software?

No. PowerLogic does not require a security device. Refer to PADS' Software License Agreement for additional information.

Is PowerLogic available on 3.5" diskettes?

No. PowerLogic is only available on CD-ROM.

Will PowerLogic be available on UNIX?

There is no current plan to make PowerLogic available for UNIX operating systems.

Installation Questions

Is PowerLogic difficult to install?

No. PowerLogic uses InstallShield, which guides you though each step of the installation. To install PowerLogic, run setup.exe and follow the prompts.

Will my existing Logic libraries and design files be overwritten?

No. PowerLogic is installed into a different directory. You can move your old files to this new area as needed.

Compatibility with Older Design and Support Files

Are schematics created in Logic compatible with PowerLogic?

Yes. You can open schematics created in Logic Version 4.0 in PowerLogic without first running a conversion program. PowerLogic also runs a database integrity check on the opened schematic file.

Are macros from Logic compatible with PowerLogic?

No. Logic's macro and menu commands are different from those in PowerLogic.

Questions on General PowerLogic Operations

How does PowerLogic differ from Logic?

PowerLogic is a true Windows95 and NT application designed for maximum ease of use and compatibility with PowerPCB.

What level of compatibility does PowerLogic have with PowerPCB?

PowerLogic and PowerPCB use a similar graphics user interface for easy transition between the two programs. They share the same parts library and library manager interface. When you read a PowerLogic net list into PowerPCB, the correct parts are automatically placed in the design file. You can also share design rules, cross-probe parts and connections, and backward annotate design changes to PowerLogic.

What new functionality does PowerLogic 3.0 contain?

Refer to the "PowerLogic 3.0 Release Highlights" document for a complete list of new functionality and features.

Is there any easier way of renaming schematic components?

Yes. Create an engineering change order (ECO) file in PowerPCB, then use PowerLogic's Backward Annotate from PCB command to update the schematic.

Does PowerLogic have a DXF translator?

No. PowerLogic Version doesn't have a conversion program to create or read DXF files.

How do I update a part on a schematic with a newer version in the parts library?

Enable Query/Modify mode, select the part, and then choose Change Type.

Why is there a small green or red dot in the upper corner of screen?

This is a system status indicator to let you know when PowerLogic is available for command input and when the system is busy.

PowerLogic uses the middle button of a three-button mouse to zoom in and zoom out. How do I do this with a two-button mouse?

Choose the Zoom icon from the toolbar, then use the left mouse button to zoom in or out. Choose the icon again to cancel zoom mode.

Chapter 2
File Operations

This chapter discusses the common file operations that you will use in PowerLogic. These commands help you to open, save, import, and export schematics. The topics in this chapter include:

Opening Files
New Files
Saving Files
Importing and Exporting Files
Creating a Netlist
ECO Operations

Opening Files

Use File/Open to open a schematic design in PowerLogic. Before displaying the File Open dialog box, a prompt is displayed to save the existing design. This occurs even if a file hasn't been opened. Choose No to display the File Open dialog box. Choosing Yes from the prompt window displays the File Save As dialog box.

File Open Dialog Box

The File Open dialog box displays the schematic files contained in the default \padspwr\logicfiles directory. The list of displayed files includes those created in PowerLogic 1.2 and PowerLogic 3.0. Choose a file, then choose Open to load the design into PowerLogic. You can choose a schematic file from a different directory using the controls at the top of this dialog box.

You can also open a file by choosing the File Open icon from the Toolbar or by dragging the file from Windows Explorer and dropping it into the PowerLogic window.

New Files

Use the File/New command to clear the current schematic from memory and start a new schematic.

1. Choose File/New from the menu.
2. The prompt *Save old file before resetting?* appears. If you choose No, a blank drawing sheet representing sheet 1 of the new schematic displays in the work area. If you choose Yes, the File Save As dialog box appears if the current schematic is untitled.

You can also create a new file by choosing New/PowerLogic Design from the pop-up menu in Windows Explorer. To create a new file this way:

1. In Windows Explorer, go to the directory where you want to create the file.
2. Choose New/PowerLogic Design from the pop-up menu. This action creates a new design file in the current directory.
3. Type a name for the file and press Enter. Make sure the file has a .sch extension. This creates a zero byte file.

When you open this file in PowerLogic, PowerLogic recognizes the zero byte file and performs a File/New using default.txt.

Saving Files

Use File/Save or File/Save As to write design information to a file. The File Save As dialog box displays the schematic files contained in the default \padspwr\logicfiles directory. The list of displayed files includes those created in PowerLogic and PADS-Logic V4.0.

The name of the currently open design is displayed in the File Name text box at the bottom of the dialog box. Choose Save to save the file. If the file name exists, a prompt is displayed to overwrite the existing file. The file name *default* is displayed for new designs.

You can also save a file by choosing the File Save icon from the Toolbar.

Importing and Exporting Files

Importing Files

The Import command inserts data from various formats into the current schematic. PowerLogic can import the following:

ASCII (*.asc)	PADS-proprietary text (ASCII) format. The ASCII files you can import includes those created from PowerLogic and PADS-Logic 4.0.
OLE, *.ole	PowerLogic allows you to embed files from other applications as OLE objects in your schematic using Edit/Insert New Object. Once you have an OLE object in your design, you can export the object as a singular item to an .ole file using File/Export. You can then import the .ole file into other PowerLogic schematics.

To import a file:

1. Choose File/Import. A prompt appears asking to save the existing design. This occurs even if a file wasn't opened.
2. Choose No to import a file. Choose Yes to save the existing design before importing a file.

   The File Import dialog box appears.
3. Choose a file, then choose Open to import the information into PowerLogic.

Exporting Files

Use File/Export to selectively extract design information from the open schematic file and save it in an ASCII format compatible with PowerLogic 1.2 and PowerLogic 3.0. You can also use Export to create default start up conditions for PowerLogic. See the "Customizing PowerLogic" topic in the "Design Setup" chapter for more information.

You can export files into the PADS ASCII format with a .txt extension.

To save all or some of the open design in ASCII format:

1. Choose File/Export from the menu to display the File Export dialog box.
2. Type a name for the ASCII file in the File Name text box. The default is the name of the currently open design.
3. Choose Save. This opens the ASCII Output dialog box. Use it to define what information to write to the ASCII file.

OLE Import/Export

PowerLogic allows you to embed files from other applications as OLE objects in your design using Edit/Insert New Object. Once you have an OLE object in your design, you can export the object as a singular item to an .ole file.

You can import an .ole file containing OLE objects into a PowerLogic schematic, and you can export an OLE object from a schematic to an .ole file.

To import an .ole file:

1. Choose File/Import. The File Import dialog box appears.
2. Choose OLE (*.ole) as the file type.
3. Locate the file you want to open.
4. Choose Open. The OLE objects import into the current schematic.

   OLE objects are placed on the sheet on which they were created. For example, if you originally placed an OLE object on Sheet 1 and a second object on Sheet 2, these OLE objects are placed on Sheets 1 and 2 when imported.

   If some of the OLE objects you want to import exist on sheets that do not exist in the current schematic, these objects are deleted. For example, you originally created OLE objects on Sheets 1, 2, and 3. The schematic you are importing the OLE objects into only has Sheets 1 and 2. The OLE objects from Sheet 3 are deleted.

To export an OLE object:

1. Select the OLE object. Handles appear around the object when it is selected.
2. Choose File/Export. The File Export dialog box appears.
3. Choose OLE (*.ole) as the file type.

4. Name the file and select a location in which to save the file.
5. Choose Save. The .ole file is created and is available for import into other PowerLogic schematics.

Note: You can also import OLE objects from their source application files using Edit/Insert New Object.

ASCII Output Options

Select Sections to Output

When the ASCII Output dialog box appears, select the items to appear in the ASCII file. Each item represents a section heading of the ASCII file. To select an item, check the box next to it. To select all sections, click Select All. To select nothing, click Unselect All. You can export the following parts and information:

Schematic Parameters	Default system settings from the Preferences tabs.
Sheet Parameters	Specific sheet information such as window scale, centering, etc.
Text	Free text together with location, level, and size.
Lines	2D line items.
Plot Parameters	Information related to the CAM output settings and configurations generated using the Print/Plot command.
Decals	Part decals and their contents.
Part Types	Library part attributes such as manufacturer, cost, and notes.
Parts	Parts used in the schematic and their reference designators.
Connections	All connections on the schematic, including paths, tie-dots, off-page flags.
Rules	Clearance, routing, and others specified in Design Rules.

Output Format

The Output Format area contains option buttons for specifying output in PowerLogic 1.2 and PowerLogic 3.0.

Filename

Displays the current path and ASCII filename.

ASCII File Format

The ASCII format files for PowerLogic and PADS-Logic V4.0 are organized into sections, beginning with a keyword enclosed with an asterisk.

The first section starts with *SCH, and gives the system set up parameters for the circuit.

The second section, starting with *CAM, lists the default settings for the plot and printing outputs.

Following this, all of the data for each sheet is grouped together in the *SHT* section. This data starts with *CAE*, listing the window zoom setting and cursor location for the sheet. Other sections are:

TXT	Free text items in the sheet.
LINES	2D line items, including library entry.
CAEDECAL	Description of the CAE symbol decals for all parts in the sheet.
PARTTYPE	Description of the part types that appear in the sheet.
PART	List of all parts, including their attributes, that appear in the sheet. Gates and connector pins are each listed separately.
BUSSES	Description of busses in the sheet, including the bus name and location.
OFFPAGE REFS	List of all off-page flags in the sheet. This includes power and ground symbols and bus connections.
TIE-DOTS	List of all tie-dots in the sheet.
CONNECTION	List of the connections in the sheet, with the signal name and path.
NETNAMES	List of all net names that are displayed in the schematic.

After all sheets are listed, the file ends with *END*.

Creating a Netlist

You can create a netlist compatible with PowerPCB Version 2.x or PowerPCB Version 3.0.

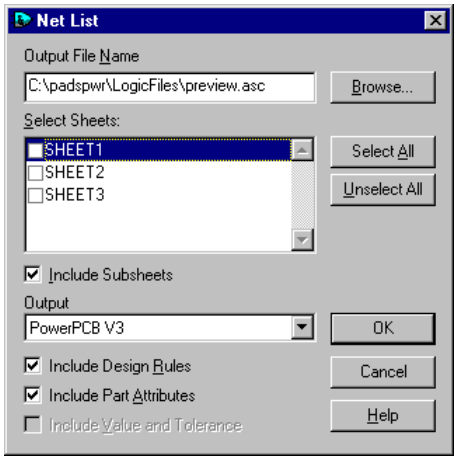
- PowerPCB V2 output can optionally include Value & Tolerance information.
- PowerPCB V3 output can optionally include Part Attribute information.
- Both formats support optional output of design rules with trace width, individual sheet selection, and hierarchical symbols inclusion.

To create a netlist:

1. Open a PowerLogic design.
2. Choose Tools/Netlist to PCB. The Netlist to PCB dialog box appears.
3. Select PowerPCB V2 or PowerPCB V3 format from the Output drop-down list box.
4. Choose from the remaining options to include other information. See the "Netlist to PCB Dialog Box" topic in this chapter for more information.
5. Optionally, choose Browse to indicate a different file name or location for the netlist. The default is the design file name with an .asc extension, saved in the \padpwr\logicfiles directory.

Netlist to PCB Dialog Box

The Netlist to PCB dialog box produces a netlist compatible with PowerPCB Version 2.x or PowerPCB Version 3.0.



Output File Name

Lists the default netlist file name, which is the design file name with the extension .asc. Type a new name or choose Browse to open the Export Netlist dialog box.

Select Sheets

Specifies which sheets to netlist. Turn on the check box next to the sheet name to include the sheet in the netlist.

Include Subsheets

When on, include in the netlist any connections to hierarchical symbols.

Output

Choose PowerPCB V2 or PowerPCB V3 format.

Include Design Rules

Includes design rules and trace width setting in the netlist for passing to PowerPCB.

Include Part Attributes

Controls whether part attributes are included in the netlist.

> Note: This option grays if you choose PowerPCB V2 format from the Output drop-down list box.

PowerPCB 3.0 and later versions now retain the attributes assigned for parts in PowerLogic.

Include Value and Tolerance

Controls whether the part value and tolerance fields for resistors, capacitors, etc. appear in the netlist.

> Note: This option grays if you choose PowerPCB V3 format from the Output drop-down list box.

When Output of value tolerance is on, the component list portion of the netlist contains the reference designation and part type name with the component's value and tolerance. For example, instead of R1, you may see:

```
R1 R1/4W,10K,5%
R2 R1/4W,100K,10%
```

PowerLogic evaluates parts that have either a different Value or Tolerance attribute as different part types. Therefore, if Output of value tolerance is on when you generate the netlist, it must be on when you perform an ECO comparison. Otherwise the comparison will report errors because it sees the part types as different.

Browse

Opens the Export Netlist dialog box. You can choose a different file name or location.

Select All

Select all sheets in the Select Sheets list box.

Unselect All

Unselect all sheets in Select Sheets list box.

ECO Operations

ECO Forward and Backward Annotation

You can export an ECO (engineering change order) file to update a PCB design with new schematic data (*forward annotation*), or import an ECO file to automatically update a schematic with data from a PCB design (*backward annotation*).

Forward Annotation

Forward annotation can be accomplished manually, using Tools/Forward ECO to PCB or automatically using Synchronize PCB from the OLE PowerPCB Connection dialog box Design tab.

Synchronize PCB compares the current PowerLogic schematic to the current PowerPCB design, then automatically updates the PowerPCB design using the resulting .eco file. This comparison uses the ECOGEN tool. See the "ECOGEN" topic in the PowerPCB On-line Help for more information.

Backward Annotation

Backward annotation can be accomplished manually, using Tools/Backward Annotate from PCB, or automatically using Synchronize SCH from the OLE PowerPCB Connection dialog box Design tab.

Synchronize SCH compares the current PowerLogic schematic to the current PowerPCB design, then automatically updates the schematic using the resulting .eco file. This comparison uses the ECOGEN tool. See the "ECOGEN" topic in the PowerPCB On-line Help for more information.

Forward ECO to PCB

Use Tools/Forward ECO to PCB to identify changes in the schematic and create an ECO (engineering change order) file that you can use to update a PowerPCB design database.

PowerLogic compares the current schematic file against another schematic file on the disk. Differences between the part type and net information in the current schematic and the file chosen for comparison are stored in the ECO file. These ECO changes define the updates that will make the original database consistent with the current schematic. This file is readable by any PADS PCB layout system and, when read in, automatically updates the PCB design database.

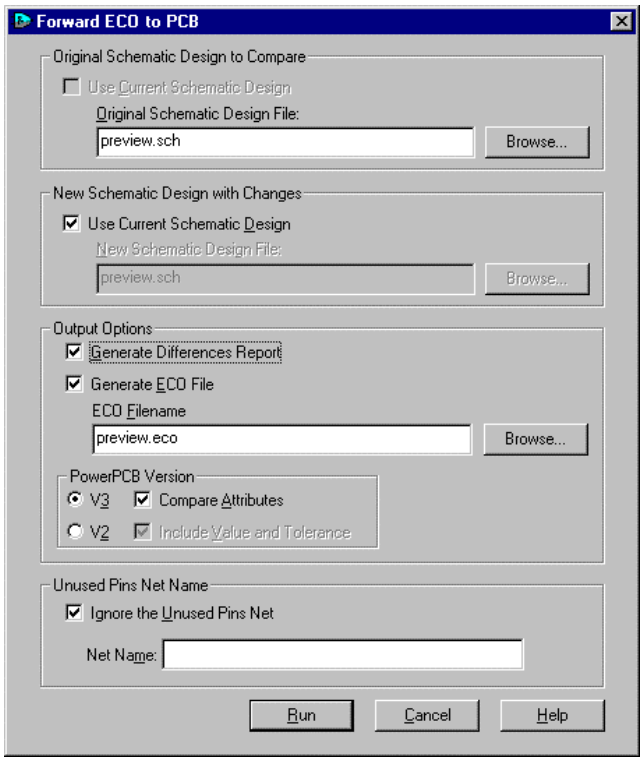
ECO changes can include:

- Added parts
- Deleted parts
- Renamed parts
- Changed part types
- Added connections
- Deleted connections
- Combined nets
- Renamed nets

To create an ECO file:

1. Choose Tools/Forward ECO to PCB. The Forward ECO to PCB dialog box appears.
2. Specify the design to update.
3. Specify the file that contains the updates that you want to place in an older design.
4. Specify an output file.
5. Specify whether the part types in the PCB design have Value and Tolerance attributes included.
6. Specify an unused pins net name if one exists in either the current or older design.
7. Choose the Run button.

PowerLogic places the ECO file in the \padspwr\logicfiles directory. You can now open the PCB layout system and import the engineering changes.



Original Schematic Design to Compare Area

Specify the schematic design to update.

Use Current Schematic Design

Turn this option on to use the open design as the original design. You cannot choose the current design as both the original design and the new design. If you do not want to use the current design as the original design, turn this option off and specify an original design in the Original Schematic Design File text box.

To compare a schematic to a design, first export the PCB file to ASCII format.

Original Schematic Design File

Specify a design to use as the original design. You can specify a .sch file, a schematic netlist file in ASCII format, or a PCB design in ASCII format. Either type a path and filename or use Browse to locate a file. If you want to use the current design as the original, turn on the Use Current Schematic Design check box.

New Schematic Design with Changes Area

Specify the schematic design file that contains the updates that you want to place in the older design.

Use Current Schematic Design

Turn this option on to use the open design as the new design that contains changes. You cannot choose the current design as both the original design and the new design. If you do not want to use the current design as the new design, turn this option off and specify a new design in the New Schematic Design File text box.

To compare a schematic to a design, first export the PCB file to ASCII format.

New Schematic Design File

Specify a design to use as the new design that contains changes. You can specify a .sch file, a schematic netlist file in ASCII format, or a PCB design in ASCII format. Either type a path and filename or use Browse to locate a file. If you want to use the current design as the new design, turn on the Use Current Schematic Design check box.

Generate Differences Report

The Generate Differences Report check box produces a report (powerpcb.rep) containing five sections:

Part differences
: This section of the report includes the reference designator and the part type for both the old and new designs. Parts that exist only in the old design are listed under the New Design column as <none>. Parts that exist only in the new design are listed under the Old Design column as <none>. Parts that have identical reference designators and part types in both designs are not listed.

Net differences
: This section lists names of the nets that do not exist under two columns: New Design and Old Design. It lists the names of the nets that do not exist in one design or the other. It also lists the nets that match, but that have different names, including nets in the old design that have been split into multiple nets in the new design. A net split operation appears as pin differences. Nets are listed alphabetically under the Old Design column, except where multiple nets are combined, where they are listed in succession. Nets that do not exist in the old design are listed at the end of this section.

Unmatched net pins in old design
: This section lists any connected pins in the old design that are missing or connected to other nets in the old design. These are the pins that are deleted from nets during the ECO process. This list provides net names in the old design followed by unmatched pins in the net. If the net does not exist in the new design, all pins in the net are listed.

Unmatched net pins in new design
: This section lists any connected pins in the new design that are missing or connected to other nets in the old design. These are the pins that are added to nets during the ECO process. This list provides net names in the new design followed by unmatched pins in the net. If the net does not exist in the old design, all pins in the net are listed.

Attribute differences
: This section of the report lists each object under the headings: Attribute Name, Old Value, and New Value. Subheadings, including object type, object name in old design, and object name in new design appear for each object in the list if different.

  Attribute differences are included only for objects that exist in both the old and new design. If an attribute is missing in either design, the value is listed as <no attr>. If the attribute exists, but has no value, it is listed as <no value>.

Generate ECO File

Allows you to specify the name of the ECO file you want to create. You can type a path and filename or use Browse to locate a file.

The ECO file contains the differences between the original and the new file in ECO commands. Once the ECO file is created, update the original design by importing this ECO file.

Compare Attributes

Restricts design comparison to only ECO-registered attributes for PowerPCB version 3 designs only. Turn off this check box to compare all attributes.

Note: Via attributes are not registered attributes and cannot be added, deleted, or changed during the ECO process.

Include Value and Tolerance

Controls whether the part value and tolerance fields for resistors, capacitors, etc. appear in the netlist for PowerPCB version 2 designs only.

PowerLogic evaluates parts that have either a different Value or Tolerance attribute as different part types.

Unused Pins Net Name area

Ignore the Unused Pins Net

Turn this option on to ignore the unused pins net in the original design, essentially preserving the existing unused pins net in the design you're updating. Otherwise, the unused pins net is deleted. Specify the net name when you turn this option on.

To compare a schematic to a design, first export the PCB file to ASCII format.

Net Name — Specify the name of the unused pins net in SPECCTRA. The default name is NOT_CONNECTED. See the "Unused Pins Net" topic in the SPECCTRA Translator On-line Help for more information. The SPECCTRA Translator On-line Help is included with PowerPCB.

Backward Annotate from PCB

Use Tools/Backward Annotate from PCB to update a schematic with changes that were made in the PowerPCB design database and saved in an ECO file. Backward annotating the changes to the schematic results in modifications of part names and pin numbers. See the "Forward ECO to PCB" topic in this chapter for information on forwarding changes from PowerLogic to PowerPCB.

You can backward annotate the following design information:

- Part changes
- Gate changes
- Pin changes
- Net changes
- Attribute changes

Note: Design Rule changes are not supported by ECO backward annotation.

After creating an ECO file in your PCB layout system:

1. Start PowerLogic and open the schematic that you want to update.
2. Choose Tools/Backward Annotate from PCB. PowerLogic prompts you to save or overwrite the existing file.
3. Type the ECO file name or choose it from the list of files.
4. Choose Open. The schematic file automatically updates. If no errors are found, the message *ECO Input Completed* appears.
5. Choose OK to continue.

Error Report File

If errors occur during back annotation, PowerLogic creates a file named eco.err. This file lists the ECO file format keyword and line number for the each error. The error report file automatically opens using the default text editor.

Part Level Backward Annotation

You can backward annotate the following part-level information.

Added Parts

A new sheet is created and all new parts are added to the sheet. Parts are placed on a grid so that parts of a medium size do not overlap. No attempt is made to avoid overlapping of larger parts.

An error message is generated if the reference designator of the newly added part already exists or if the part doesn't exist in the Library.

Note: The part is not added to the schematic if the reference designator already exists.

If the part contains Signal Pins, these pins are included in the add pin function. Backward annotation does not currently support signal pins so an error message is created.

Changed Parts

If the changed part is a multigate part, all gated are updated to the new part type.

An error message is generated if the new part does not exist in the design or in the Library, or if the gate or pin count is incompatible.

Deleted Parts

If the deleted part is a multigate part, all gates are deleted.

An error message is generated if the part is still connected to a net or the part does not exist.

Reference Designator Name

If the part being renamed is a multigate part, all gates are updated. An error message is generated if the old reference designator does not exist.

Gate Level Backward Annotation

You can backward annotate the following gate-level information.

Swapped Gates

PowerLogic creates an offpage symbol at each swapped gate.

An error message is created if the gate does not exist.

Pin Level Backward Annotation

You can backward annotate the following pin-level information.

Swapped Pins

PowerLogic creates an offpage symbol at each swapped pin.

Pins Added to a Net

A pin can be added only if it is not already connected to another net. If the pin is a gate pin (a visible terminal pin on the gate symbol), an offpage symbol is created.

An error is created if pin is already connected or the pin is a signal pin already assigned to a net.

Pins Disconnected from a Net

If the pin is a gate pin, the connection is deleted if it connects to a tie-dot or offpage symbol. If the connection goes to another gate pin, the connection is broken and, an offpage symbol is added.

This command generates an error message if the pin is not connected to the net in question.

Net Level Backward Annotation

You can backward annotate the following net-level information.

Joined Nets

The first net is renamed to be the same name as the second net.

Nets Created by Splitting an Existing Net

A Delete Pin from Net operation is performed to remove them from the existing net, followed by an Add Pin to Net operation to add the pin to the new net.

Renamed Nets

All subnets of the old net on all sheets are renamed. If any of the subnets contain Power or Ground symbols without netnames, netnames are added to these symbols.

An error message is created if the new net already exists.

Attribute Level Backward Annotation

You can backward annotate the following attribute-level information.

New Attributes

A new attribute in a part updates all parts of the same type. If the attribute name does not exist, it is added with the assigned value.

An error is created if the part does not exist.

Note: Unsupported attribute types, such as net or net class are ignored.

Deleted Attributes

Deleting an attribute for a part-type deletes the attribute on all parts of that type in the design.

An error message is generated if the part or attribute name does not exist.

If the attribute command specifies an object type not supported for general attributes, such as net or net class, the attribute command is ignored.

ECO File Format

The format used is similar to the PADS layout program's ASCII format. Each type of data begins with a header line with a key word surrounded by asterisks (*). The first line of the file is:

```
*PADS-ECO*
```

The end of the file (EOF) entry is:

```
*END*
```

Add remark lines with the entry:

```
*REMARK*  remark information etc.
```

These are the available ECO commands:

Add a pin to net

The command starts with:

```
*NET*
```

A line indicating the net to which the pin is added follows this line:

```
*SIGNAL* netname 10
```

where netname is the net to add the pins to and 10 is the trace width to associate with the connections. If the net name does not currently exist in the design it will be added. This is followed by the pin(s) to add to the net

ref1.pin1 ref2.pin2

Add a part

The command is:

```
*PART*
```

The format for the part entry is:

```
refdes parttype
```

where refdes is the part reference name and parttype is the part type name. When parts are added in the PCB, they will be placed at system origin of 0,0.

Join two nets together

The command starts with

```
*JOINNET*
```

This is followed with the line indicating the nets to join:

```
OLDNET0 OLDNET1
```

where OLDNET0 and OLDNET1 are the names of the nets to combine. The new combined net uses the name OLDNET1. A connection is added between the nets using two random pins in the selected nets. The trace width of the added connection is the same as that of a connection in the first net (OLDNET0).

Delete a part

The command is:

```
*DELPART*
```

The format for the line is:

```
refdes parttype
```

where refdes is the part reference name to delete and parttype is the part type name. (The part type name is not required.) All pins of the part to be deleted must already have been disconnected from the connection nets in the design or an error will be reported.

Delete a pin from net

The command starts

```
*DELPIN*
```

This is followed by a list of pins to delete from each net, in the form:

```
refdes.pinnumber signame
```

where refdes is the part reference name, pinnumber is the pin number to disconnect, and signame is the net of which the pin is currently part.

Change a component's part type

The command is:

```
*CHGPART*
```

The format for the change part line is:

```
refdes oldparttype newparttype
```

where refdes is the reference name of the part to change, oldparttype is the old part type, and newparttype is the new part type.

Splitting a net into two new nets

The command starts:

```
*SPLITNET* SPLIT NET INTO TWO NEW NETS
```

two lines follow, listing the new signal names and the pins:

```
*SIGNAL* oldsigname
ref1.pin1 ref2.pin2
*SIGNAL* newsigname1
ref3.pin3 ref4.pin4
```

where the pins following the *SIGNAL* statement are in the first net, and those following the second line are in the second net.

Rename a part

The command is:

```
*RENPART*
```

The format for the rename line is:

```
oldrefdes newrefdes
```

where oldrefdes is the old name and newrefdes is the new name. To facilitate the renaming operation, duplicate name checking is not run until after all rename information has been read. This allows the above rename list to run without a conflict. If any error is encountered, no parts in the list are renamed.

Rename a net

The command is:

```
*RENNET* RENAME NET
```

The format for the rename net entry is:

```
oldname     newname
```

where oldname is the old net name and newname is the new net name.

Swap a gate

The command is:

SWPGATE GATE1 GATE2

Swap a pin

The command is:

SWPPINS REFDES PIN1.PIN2

Chapter 3
Design Setup

This chapter discusses how to set up and customize PowerLogic to fit your work style. As part of customizing PowerLogic, you will also learn how to use macros to help you work faster. The topics in this chapter include:

Setting Preferences
Setting Colors
Customizing PowerLogic
The Macro Recorder

Setting Preferences

Using Preferences, you can preset options for commands in PowerLogic, set up the PowerLogic commands, and override the default settings in the default.txt file. Setting Preferences also allows you to set up a working environment that best suits your design and the way you work.

1. Choose Preferences from the Setup menu. The Preferences dialog box appears.
2. Choose the tab that represents the PowerLogic area you want to set preferences for. Choose from Global, Design, or Heights/Widths.
3. When you choose the tab, the options for that tab appear. Set the options you want to change.
4. Choose another tab to set preferences for.
5. When you finish setting preferences, choose OK. The changes are saved and the Preferences dialog box closes. Choose Cancel to close the Preferences dialog box without saving any of the changes you make.

Global Tab

The Global tab sets interface parameters and attributes that are global to the design process regardless of the current editing mode.

Bitmap Size X and Y

Sets the size of the bitmap area in the Status Window. The bitmap area is the area of the board that PowerLogic can scroll across at any given zoom level. Type a value in pixels.

Use Bitmap

Turns the bitmap area in the Status Window on or off. Turn it off to save memory or speed screen redraws. If you turn the bitmap area off, use the middle mouse (zoom) commands or the Status Window to scroll around the work area.

Cursor Style

Displays the cursor as an arrow (normal), small cross, large cross, or full screen crosshair. Choose a cursor style from the drop-down list box.

Diagonal Cursor

Displays the cursor diagonally.

Keep View on Resize

Maintains the proportional scale and the same area of the screen in the window when you resize the view.

Display Dot Grid

Defines the spacing for the dot grid display in the work area. This grid is a visible guide for drawing lines, decals, connections, etc. It is independent of the system grid on the Status Bar.

To change the display grid, enter the new value in units of 2 mil increments. If the grid is not visible when the work area redraws, you probably set an interval that is too small to display at all zoom levels. Zoom in several times to display the grid. A reasonable value for the display grid is 500 or 1000.

Real-time Redraw

Displays items one by one as they redraw, rather than all at once. When this check box is off, your screen pauses until all objects are ready to redraw. Turning Real-time Redraw off is the faster redraw setting.

Real Width

Defines the minimum width to display text, 2D lines, decal outlines, in addition to connections with their actual width. Lines with a width smaller than this value appear with only a center line to save memory and speed redraws. If you set this to zero, all lines appear at their actual width. Larger values speed up redraws. Type a value in current design units.

Display OLE Objects

When on, displays linked and embedded objects in the work area.

Update on Redraw

Updates the PowerLogic linked or embedded object in the container application when the following is true:

- You are editing the PowerLogic object in a separate window.
- You choose the Redraw icon in the separate window.

To increase performance, turn this option off. This option only effects PowerLogic when it is embedded in another application.

Draw Background

Draws the PowerLogic background color in the linked or embedded PowerLogic object. You set this color in the Display Colors Setup dialog box. When this option is off, the background of the PowerLogic object is transparent: you can see the through the object to the container application's background.

This option only effects PowerLogic when it is embedded in another application.

Backup Interval

Defines the time in minutes between automatic backups. Type a value between 1 and 30 minutes.

Number Backups

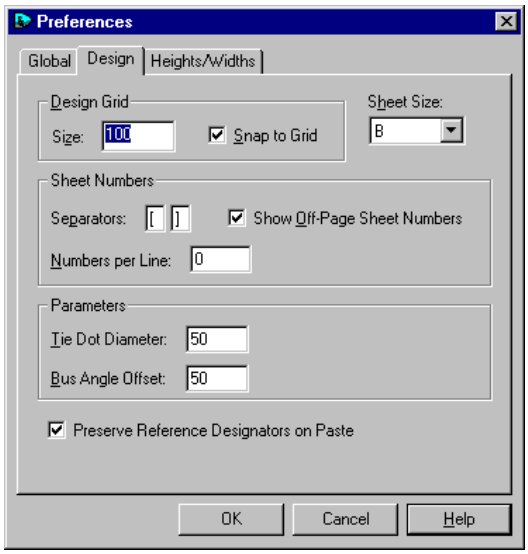
Sets the number of sequential, different backup files to create. Create between 1 and 9 different and sequential backup files. PowerLogic increments and numbers the files as logic00.sch, logic01.sch, etc.

Backup File

Opens the Backup File dialog box where you can choose a location and filename for the backup file. The default backup filename is logic00.sch which PowerLogic saves to the \padspwr\logicfiles directory. If you change the backup filename, PowerLogic automatically adds the numeric suffix.

Design Tab

The Design tab defines the parameters for schematic editing.



Design Grid Size

Sets the snap grid size. The snap grid is the minimum distance for cursor movement. Type a value between 2 and 1000 that is a multiple of 2, for example, 2, 4, 50, or 100.

Snap to Grid

Specifies that an item/object snaps to the grid during editing.

Sheet Size

Sets the sheet size. When you change the sheet size, it affects all sheets of the schematic. Choose a size from either US standard sheet sizes A through F or international standard drawing sizes A4 through A0.

Sheet Numbers Separators

Sets the open and close characters that enclose sheet numbers. Type characters such as quotes " " and curly { } or straight [ ] brackets.

Show Off-Page Sheet Numbers

Displays sheet reference numbers adjacent to off-page reference symbols and bus names.

Numbers per Line

Sets how many sheet numbers display per line when Show Off-Page Sheet Numbers is on. The sheet numbers display on a line below the net name. Type a value between 0 and 99. Depending on the value you specify, you can stack or line up the sheet number references.

Tie Dot Diameter

Sets the diameter of tie dots. Type a value between 0 and 100.

Bus Angle Offset

Defines the starting point distance from the bus for the bus tap. The bus tap joins connections to the bus. Type a value between 0 and 250.

Preserve Reference Designators on Paste

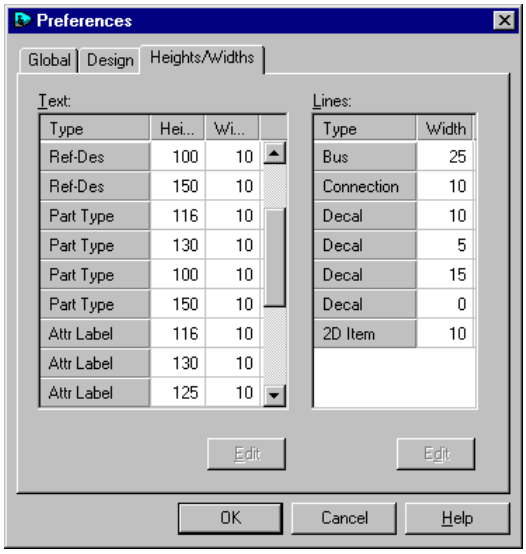
Uses a group's current reference designator assignments when possible. See the "Preserving Reference Designators" topic in the "Design Operations" chapter for more information.

Turn this option off to assign reference designators in the pasted group the first or next available number

Heights/Widths Tab

The Heights/Widths tab specifies the height and/or width of text characters and lines. When you change text or line size, PowerLogic modifies all entries/objects in the schematic with the new values.

Changes you make to symbol text and decal line sizes only modify the values for the current schematic, not the part type in the library. To change the part type in the library, you must modify it using the Part Editor.



Text Area

Lists the different text height and width settings for all occurrences of net names, pin numbers, pin names, reference designators, part type names, attributes, and free text items. As you add or modify text strings for these items, new entries are added to the Type column.

You can change a height or width setting in the Text multicolumn list box and update all text strings that use those values.

Height	The character height. Type a value in the range 10-1000.
Width	Specifies the line width of the character font, Type a value in the range 1-50.

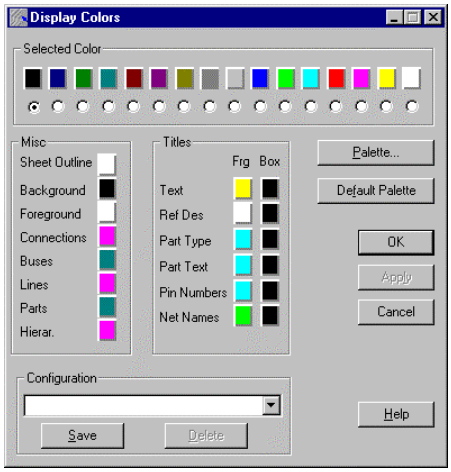
The text height definition defines the total distance from the top of capital letters to the bottom of lowercase letters, such as y. The font allocates 25% of the text height to the lower portion of the letter; therefore, an uppercase character with a value of 100 mils is actually 75-mils-high.

Lines Area

Sets the line width for connections, busses, and the lines within logic symbols and 2D items. Type a value between 1 and 250 for connections and busses. You can set multiple widths for decals and 2D items. Type a value between 1 and 250 for decals and 2D items.

Setting Colors

To open the Setup Colors dialog box, choose Setup/Display Colors. PowerLogic saves the setup for the schematic with the schematic. The color selection for display does not affect plotting of the schematic.



Selected Color Area

Sets a color to assign to items. To change the available colors, choose Palette.

Miscellaneous Area

Applies colors, including foreground and background, to the following items:

- Sheet Outline
- Background (work area surface)
- Foreground (dot grid and highlights)
- Connections
- Busses
- Lines
- Parts
- Hierarchical components

To make an item invisible, set it to the Background color. Do not set the Foreground color and Background color to the same color.

Titles Area

Applies colors to the following text items:

- Text
- Reference Designators
- Part Types
- Part Text
- Pin Numbers
- Net Names

Two color box columns appear next to the text item list. Box indicates the color of the box drawn around the text. This box indicates the exact size of the text item when it is plotted, thereby helping you avoid overlaps while moving the item, and it provides visibility of the text item at small zoom levels.

Configuration Area

Saves color assignments to a file. When you finish setting color assignments, choose Save from the Configuration area. Type the configuration name in the Save Configuration dialog box. PowerLogic saves configuration files in the \padspwr\powerlogic directory with a .ccf extension. The configuration names also appear at the bottom of the Setup menu.

To modify an existing configuration with the current color settings, choose Save. The current configuration filename appears in the Save Configurations dialog box. You can overwrite the settings in this file or specify a different .ccf file, and choose OK.

Palette

Opens the Color dialog box where you can specify new colors or customize colors that appear in the Selected Color area.

Refer to the *Microsoft Windows User's Guide* for more information on changing the Color Palette.

Default Palette

Restores the default color settings in the Selected Color area.

To change the default palette, complete the modifications to the Selected Color area and choose Save. Type default in the Save Configurations dialog box to override the existing default configuration and choose OK.

Customizing PowerLogic

There are several files in the \padspwr\powerlogic directory that PowerLogic uses to define default operation. You can customize the user interface for PowerLogic by modifying system settings and saving the files. The most important file is default.txt. This ASCII file is read into memory when you start PowerLogic or choose File/New to begin a new design. The list of files used for customizing PowerLogic are:

- default.txt
- width.dat
- future.dat

default.txt

This file is a PowerLogic ASCII file. The settings in this file are used every time you start PowerLogic or start a new schematic; it contains default preference settings. To change the default start-up conditions for PowerLogic:

1. Choose Setup/Preferences to open the Preferences dialog box.
2. Change the tab settings to your preferred work environment.
3. Choose OK to save the changes.
4. Choose File/Export, and then use the controls at the top of this dialog box to set the directory to \padspwr\powerlogic.
5. Choose default.txt from the list of files and answer Yes to the prompt to overwrite the existing file. Type the name in the File Name area if it does not exist.
6. Choose Save. The ASCII Out dialog box appears.
7. Choose Select All, then choose Save.

The next time you start PowerLogic or when you choose File/New, your customized settings are used.

width.dat

This file defines the trace widths for nets when PowerLogic creates the net list. You can define specific net names with their associated trace width. You can also assign a default net width for all other signals.

When PowerLogic creates a netlist, this file is read. If a net name appears in this list, it overrules any other trace width definition for the net in the schematic or in the library.

future.dat

This file is useful if you intend to output your schematic in the Futurenet netlist format. You can define the Futurenet attribute number cross-reference for each part attribute in the circuit.

Attribute numbers 1 through 10 are preassigned in Futurenet. You can assign others. The file format is:

```
PADS-attributename:Futurenet-Attribute-no
```

The PADS-attributename parameter must appear exactly as it appears in the library (for example, Partname: is not the same as Part Name:).

The Futurenet-Attribute-no. is an integer between 1 and 100.

The following is a sample future.dat file:

```
VALUE:              4
TOLERANCE:          6
STRESS:             7
PCB DECAL:          80
PART Description:   81
Part number:        82
Part Desc.          81
$:                  84
Mfg. #1:            85
Mfg. #2:            86
Spice model:        87
Spice qualifier:    88
Spice Pin order:    89
Spice Par1:         90
```

The Macro Recorder

A macro is any combination of commands, keystrokes, and mouse clicks that you record to replay as a single action. You can record virtually any set of procedural steps in PowerLogic for replay, thereby simplifying redundant activities. In this way, macros are another method of customizing PowerLogic.

> Note: Macros are created and stored in files called macro files that have a .mcr extension.

The PowerLogic macro recorder is based on the design database grid rather than the screen display. Recorded actions are screen resolution independent, allowing you to create macros on one system and play them back on other platforms and systems with different screen resolutions.

Start a macro command by selecting Setup/Macro. The Macro dialog box appears, which provides access to all of the macro command functions.

The Macro command also allows you to record an entire PowerLogic session in one .log file, providing troubleshooting information and documenting failures.

Macros and Dialog Boxes

Dialog box actions are recorded as results rather than actions, so when you replay, you don't see the dialog boxes in the replay process as you do with the Microsoft Windows macro recorder. Because of this, you can't create a macro that stops on an open dialog box; it must follow through to some result or action.

For example, you can create a macro that chooses File/Open, selects a file, and chooses OK. The macro, when played back, opens that file. You cannot, however, record a macro that selects File/Open, opens the Open dialog box, then waits for you to select a file. The Microsoft Windows macro recorder performs like this; however it is screen-dependent and you cannot transfer its macros from one computer to another.

Macro Files

Macros are created in and stored in macro files that have a .mcr extension. The Macro command allows you to make collections of design specific macros by creating different macro files. You can create design specific macros because a set of macros for design A may be irrelevant to design B. Macro files are stored in the \padpwr\logicfiles directory.

To access a macro you created, you must first Open the macro file. To create a new macro file, choose New from the Macro dialog box.

Macros are saved to the open macro file immediately after you record them (when you choose the Stop icon). See the "Creating a New Macro" topic in this chapter for information on creating new macros.

The last macro file opened is the default file and automatically opens when you start PowerLogic. PowerLogic tracks which macro file was last opened in the [MACROS] section of the pwrlogic.ini file.

When you open a macro file through the Macro dialog box, existing macros are listed in a text window. You can then select individual macros to run via the Run command. You can add macros to and start macros from the menu using shortcut keys. See the "Assigning Shortcut Keys" topic in this chapter for more information.

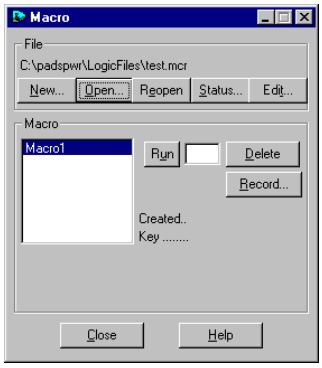
Macro File Format

Macro files are stored in ASCII format so you can Edit them. When you edit a macro, it loads into the default text editor. If you edit a macro, the ASCII file changes and you must reopen it in PowerLogic for those changes to take effect. A command called Reopen exists to open the last macro file.

Macro Dialog Box

The Macro dialog box provides access to all macro functionality. This includes creating new macro files, opening/editing existing macro files, modifying existing macros, and accessing the macro record functions.

New macros are, by default, named MACRO#, where # is a numeric sequence such as macro1, macro2, macro3, etc. To provide a specific macro name, delete the default and type a new name for the macro. You can use any combination of letters and/or numbers without spaces in a macro name. The Macro Name list box in the Macro dialog box displays names up to 18 characters long. See the "Creating a New Macro" topic in this chapter for more information.



New

Creates a new macro file. The Create Macro File dialog box appears, providing filename, directory, drive, and extension settings.

> Note: After creating a new macro file, use the Open command to access the file.

Open

Opens an existing macro file. A File Open dialog box appears, displaying the default directory and files with the default .mcr extension.

Reopen

Opens a file you previously edited.

Status

Opens the Macro Status dialog box where you can assign or change shortcut keys and where you can add macros to or delete macros from the menu. See the "Assigning Shortcut Keys" topic in this chapter for more information.

Edit

Loads the macro file in the text editor that you chose during PowerLogic installation to view or manually edit the macro file. If you chose Notepad as the default text editor, longer macro files may not load because of size constraints in Notepad.

Accessing large files via the Edit function requires that you install an ASCII text editor with a suitable file size capacity; modify the [GENERAL] section of the pwrlogic.ini file, and specify a new text editor executable name. Include the drive and directory if the new editor is not in your Windows directory.

Note: Editing the macro file or pwrlogic.ini is an advanced operation; generally, you should not edit macro files because you can access all macro operations through the Macro dialog box.

Macro Name

Displays the new macro name. Macro Name also displays any macros created for the open macro file. If a description of the macro was entered during creation, it appears directly below the Macro Name list box when the macro is highlighted. Any assigned shortcut keys appear next to this list box.

Run

Starts an existing macro. First highlight the macro from the Macro Name list box. You can repeat a macro a specified number of times by entering the number in the Repeat Count text box located to the right of the Run button. See the "Macro Playback" topic in this chapter for more information on starting macros.

Repeat Count

Starts the highlighted macro a specified number of times. To enter the value, type the number of times you want to repeat the macro in the text box. Choose Run to play back the macro.

Delete

Removes the highlighted macro from the macro file.

Record

Creates a new macro in the open macro file. See the "Creating a New Macro" topic in this chapter for more information.

Creating a New Macro

To create a new macro, select Tools/Macros to open the Macro dialog box. Before recording a new macro, you must Open an existing macro file or create a New macro file.

New macros are given a default name of MACRO#, where # is a numeric sequence such as macro1 or macro2. You can name each macro. To help identify the functions of macros, the Recorder provides an area to describe each created macro.

After establishing the macro file, start the Recorder as follows:

1. Choose Record. The Record dialog box appears.

2. Type a macro name that consists of letters and or numbers without spaces. The Macro Name list box can display names up to 18 characters long.
3. Type a description for the macro. This describes the macro's function when you highlight the macro name in the Macro dialog box.
4. Choose any of the Record mode options:

   Mouse Compression mode
   Compresses mouse movements. See the "Compress Mouse Mode" topic in this chapter for more information.

   Record Prompt mode
   Records prompt windows that are encountered in a macro. See the "Record Prompt Mode" topic in this chapter for more information.

   Coordinate mode Records mouse coordinates. See the "Coordinate Mode" topic in this chapter for more information.

5. Choose OK to record the macro. The Record Mode toolbox appears.

6. Perform the sequence of steps you want to record.

You can choose the Pause icon from the Record Mode toolbox to create a pause during playback. When the macro pauses during playback, a dialog box with a standard message appears. You can edit the macro file if you want to change the message.

7. Choose the Stop icon from the Record Mode toolbox to stop recording. The macro is automatically saved to your open macro file. You return to the Macro dialog box.
8. Define a shortcut key. See the "Assigning Shortcut Keys" topic in this chapter for more information.
9. Turn Add to Menu on if you want to add the macro name to the Tools menu. See the "Assigning Shortcut Keys" topic in this chapter for more information.

Note: Before you can add a macro to the menu, you must first assign a shortcut key.

Compress Mouse Mode

The Compress Mouse Mode icon sets whether mouse movements are recorded or compressed. Compression is recommended under most circumstances because it significantly reduces the size of your macro file. Recording mouse movements increases the file size, but documents coordinate information if required for a special application.

Mouse movements are not recorded.

Mouse movements are recorded.

Record Prompt Mode

The Record Prompt Mode icon sets whether to record prompt windows. Prompt windows confirm actions. When Record Prompt mode is off, you must enter responses manually during playback of the macro.

Prompts are recorded in the macro.

Prompts are not recorded in the macro.

Coordinate Mode

The Coordinate Mode icon sets recorded mouse coordinates for relative coordinates, distance from last pick point, or absolute coordinates (all distances from 0,0 coordinate location).

Records coordinates relative to last selection point.

Records coordinates from absolute 0,0 location.

Assigning Shortcut Keys

It is possible to start a macro without opening the Macro dialog box by assigning a combination of keystrokes called shortcut keys to each macro. Use modifier keys, Ctrl and Shift, and function keys, F1 through F12 to create the key combination. The Macro Status dialog box assigns or changes shortcut keys and adds macros to or deletes macros from the Tools menu. Open the Macro Status dialog box by choosing Status in the Macro dialog box.

To assign shortcut keys:

1. Choose a key assignment from the list box in the Assigned Keys area.
2. Choose a macro from the drop-down list box in the Assigned Keys area. The macro name appears next to the key combination.
3. Optionally, turn In Menu on to add the macro to the menu.

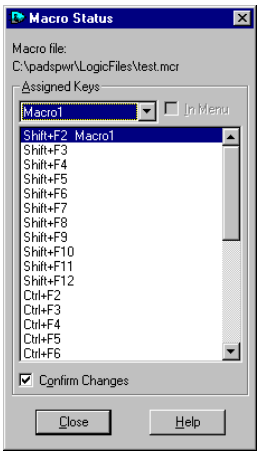
Note: You can change the shortcut key assignment of a macro following the same steps, but you cannot use the same shortcut keys for more than one macro. If you try to assign the same shortcut key to more than one macro, a prompt indicates that the shortcut key will be reassigned.

You can only assign shortcut keys to the open macro file.

To remove any key assignment from the macro:

1. Highlight the current key assignment.
2. Enter the list box to display the macros.
3. Choose the empty space displayed above the first macro.

Macro Status Dialog Box



Drop-down list box

Lists all of the macros in the open macro file.

In Menu

Adds macros to or removing macros from the Tools menu. Files added to the menu appear next to the assigned macro name.

List box

Lists all shortcut keys. Current assignments of macros and additions to the menu, In Menu, are also reflected in this list.

Note: To add a macro to the menu, you must first assign a shortcut key.

Confirm Changes

Confirms changes to existing assignments.

Macro Playback

There are three methods available for playing back macros contained in the open macro file:

- Choose Tools/Macros from the menu, highlight the macro to play from the list, then choose Run.
- For macros that you added to the Tools menu with Add to Menu, choose Tools and select the macro name under Macros.
- Press the assigned shortcut key.

See the "Assigning Shortcut Keys" topic in this chapter for information on assigning macros and adding macros to the Tools menu.

Recording a PowerLogic Session

When you enable on-line macros, you create a .log file of the session which you can play back. Use .log files to reproduce problems for program troubleshooting or to start a series of actions to create items in the design.

To enable recording of an entire session, change the following in your pwrlogic.ini file:

```
[macros]
LOGMode=0
LOGCompressionMode=1
```

to

```
[macros]
LOGMode=1
LOGCompressionMode=0
```

The file next.log is created in your \padspwr\powerlogic directory the next time you start PowerLogic. All actions in the next session are recorded to this file, and the file is stored when you exit PowerLogic or a system failure occurs. In addition to the .log file, a copy of the current pwrlogic.ini is created and named next.ini.

Note: Existing next.log and next.ini files are overwritten each time you start PowerLogic. Copy these files to other names to retain them.

Play Back a Recorded Session

To play back a recorded .log file:

1. Rename the current powerlogic.ini file to a temporary name such as pwrini.sav.
2. Rename next.ini to powerlogic.ini.
3. Start PowerLogic.
4. Choose Tools/Macros, choose Open, and change the file type to .log.
5. Locate the .log file you want to run and choose Run.

When you finish analyzing a recorded session, remember to restore the original pwrlogic.ini; the one you renamed in Step 1.

Submitting a .log File to PADS

Whenever you submit a problem to PADS Technical Support, it is useful to submit a copy of the next.log and next.ini files that reproduce the problem. Also, include the design file so Technical Support can accurately reproduce the anomaly.

Chapter 4
Design Operations

This chapter discusses general design operations, from adding and editing parts, creating and modifying hierarchical symbols, and adding connections to setting up sheets. This chapter also discusses the many different editing modes available to you in PowerLogic. The topics in this chapter include:

Starting a New Design
Design Toolbox
Bus Toolbox
Sheets
Modes of Operation
Group Operations
Cut, Copy, and Paste
Query/Modify

Starting a New Design

When you start PowerLogic or choose File/New from the menu, a drawing format is automatically added to the work area. See the "Design Tab" topic in the "Design Setup" chapter for information on changing the default drawing format. Also use Preferences to set the default working grid, display grid, text size, etc.

To add new design information, open one of the toolboxes in the toolbar:

- Use the Design toolbox to add and edit parts from libraries, create and modify hierarchical symbols, add connections, and swap pins and reference designators.
- Use the Bus toolbox to define bus structures.
- Use the Drafting toolbox to create and edit non-electrical information, including text, charts, notes, etc. See the "Drafting Operations" chapter for more information.

Design Toolbox

The Design toolbox adds and edits libraries parts, creates and modifies hierarchical symbols, adds connections, and swaps pins and reference designators.

Choose Window/Design Toolbox or choose the Design Toolbox icon from the toolbar to open the Design toolbox.

	Add Part	Adds a library part to the schematic. See the "Adding Parts" topic in this chapter for more information.
	Add Connection	Adds a new connection. See the "Adding Connections" topic in this chapter for more information.
	Edit Part/Hierarchical Symbol	Invokes the Part Editor to modify a library part or a hierarchical symbol. See the "Modifying a Hierarchical Symbol" topic in this chapter for more information.
	New Hierarchical Symbol	Creates a hierarchical symbol either bottom-up (underlying logic defined) or top-down (underlying logic undefined). See the "Hierarchical Design" topic in this chapter for more information.
	Swap Reference Designator	Exchanges the selected reference designators. See the "Swapping Reference Designators" topic in this chapter for more information.
	Swap Pins	Exchanges the selected pins. See the "Swapping Pins" topic in this chapter for more information.

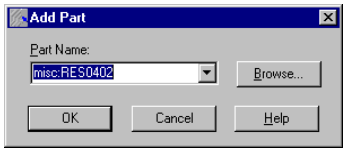
Adding Parts

Use the Add Part icon to load a part from a library into the current schematic drawing. PowerLogic automatically assigns a reference designator when you add the part. See the "Adding Single Gate Parts" and "Adding Multigate Parts" topics in this chapter for more information.

See the "Copy Mode" topic in this chapter for information on adding parts by making copies of parts that already exist in the schematic.

To add a part from the library:

1. Choose the Add Part icon from the Design toolbox, or choose Add Item/Add Part from the pop-up menu. The Add Part dialog box appears.



Part Name	Specifies the part that is identified for adding. To use a different part, type the full part name.
Browse	Opens the Get Part-type from Library dialog box for library searching.

2. Type a part or choose one from the drop-down list box.

   If the list does not contain the part you need, you can enter a part prefix followed by the asterisk character then choose Browse to retrieve the part from the available libraries. You can also just choose Browse and use the filter in the Get Part-type from Library dialog box to locate the part. Use the wildcard convention, with or without leading characters, to expand or narrow the filter. Choose Apply to activate the filter. See the "Get Part-type from Library Dialog Box" topic in this chapter for more information.

3. Choose a part and choose OK. The part attaches to and follows the cursor movement.

   Optionally, you can rotate or mirror the item before you indicate its location. Choose a pop-up menu command as often as needed to set the correct orientation of the item. See the "Add Part Pop-up Menu" topic in this chapter for more information.

4. Indicate the part location. After you place one part, another instance attaches to the cursor.

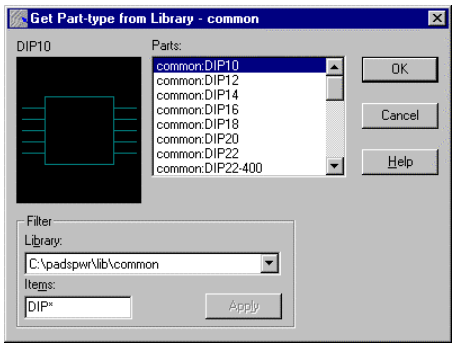
   Optionally, you can add multiple copies of the item after the initial part location is indicated. Choose Step and Repeat from the pop-up menu. See the "Step and Repeat" topic in this chapter for more information.

5. When you finish adding parts, choose Cancel from the pop-up menu.

The sixteen most recently used parts are available in the Part Name drop-down list box. You can clear this buffer by removing the entries in the powerlogic.ini file, under the [Last Added Parts] heading.

Get Part-type from Library Dialog Box

When you choose the Browse button from the Add Part dialog box, the Get Part-type from Library dialog box appears.



Parts

Displays the parts that PowerLogic matched to the search filter settings. Choose the part you want to add from this list. The decal of a selected part displays in the area to the left of the list box.

Filter area

Searches the selected library for a specific part or item name or for names that match a wildcard text string. See the "Preface" for more information.

Add Part Pop-up Menu

Rotate 90 — Toggles between a rotation of 90 degrees counterclockwise and the original orientation.

X Mirror and Y Mirror — Reverse the pins while placing a part. This is often useful for improving the flow of the schematic.

X Mirror mirrors the part in the X direction. When selected, the part flips about a vertical reference line. Part text is not mirrored. Y Mirror mirrors the part attached to the cursor, in the Y direction; that is, flips the part about a horizontal reference line. Part text is not mirrored.

Next Type — Selects the gate with the electrical properties that you require. When you add a part with more than one gate type in the part, Next Type allows you to select the next type of gate in the part.

Alternate — Places an alternate symbol, if available.

Adding Single Gate Parts

Single gate parts are parts that are represented by a single schematic symbol. When you add the part to the schematic, unique reference designator assignment starts with the next available reference designator number. The "Adding Parts" topic in this chapter lists the step-by-step procedure.

Adding Multigate Parts

Multigate parts are parts that consist of more than one symbol or gate to represent a complete part; for example, a 7400 which contains 4 gates. See the "Adding Parts" topic in this chapter for the step-by-step procedure for adding multigate parts.

When you add a new multigate part, the gate with the lowest set of pin numbers is used along with the next available reference designator and an alphanumeric suffix, or gate modifier. The reference designator and suffix are separated by the dash (-) character. For example, a 7400 added to a new schematic would use the gate with pins 1 through 3 and would be assigned as part U1-A. When you add additional gates of the same part, the next available gate and gate modifier is used, for example, the gate with pins 4 through 6 is added and assigned as U1-B.

PowerLogic continually manages the packaging of gates in a schematic. If you delete, rename, or change the part type of a gate, PowerLogic keeps track of these changes and fills in the gaps when a new gate is added.

Using Alphanumeric Prefixes

You can use any combination of letters and numbers in PowerLogic to define a reference designator prefix. You specify this prefix in the General tab of the Part information dialog box when you create a part.

If you specify a reference designator prefix that contains alphanumeric characters, PowerLogic maintains this prefix for multigate parts. A prefix of U1A for a part containing four gates, when added to a design, would use reference designator assignments U1A-A, U1A-B, U1A-C, and U1A-D.

Adding Connector Pins

Connector pins are special symbols within PowerLogic libraries. You add them like multigate parts whereby the pin number increments each time you add a connector symbol. PowerLogic does not increment the reference designator until you use all the connector pins.

After adding one or more pins of a connector and establishing a reference designator, use Copy mode to add additional pins. This causes PowerLogic to use the same reference designator for new pins.

Using Alternate Symbols

Some parts can appear in the schematic with more than one representation. One example of this is the Nand gate with its DeMorgan representation. A second example is the use of IEEE and ANSII symbol standards. Within PowerLogic's library, you can define one primary and three alternate symbols for a part. While placing parts in the schematic, use the Alternate command to select the version you want to use in the schematic. You can also use power symbols, ground symbols, and off-page reference symbols as alternate symbols.

When you add a new part or copy an existing part, and you want the alternate representation, choose Alternate from the pop-up menu before placing the part.

To change an alternate, choose Alternate from the pop-up menu during Copy or Move mode.

Adding Connections

When you add connections they must have electrically meaningful start and end points. Start a connection on either a symbol pin or another connection. End a connection on a symbol pin, another connection, a bus, an off-page reference, or a ground or power symbol. PowerLogic prevents you from ending a connection just anywhere in the database. This eliminates time-consuming post-design error checking for dangling connections.

To add a new connection:

1. Choose the Add Connection icon from the Design toolbox, or choose Add Item/Add Connection from the pop-up menu.
2. Indicate the starting point of the connection. The connection follows the cursor movement. You can choose Angle from the pop-up menu to specify diagonal segments in the connection.
3. Indicate corners as needed. You can choose Del Corner from the pop-up menu to back up to the previous corner if necessary.
4. A connection cannot end in open space. Locate the cursor and double-click to end the connection.

To end a connection with a power, ground, or off-page reference symbol, choose the appropriate symbol from the pop-up menu during the last step of this procedure. If you end the connection on a bus, see the remaining steps in the "Adding Connections to a Bus" topic in this chapter.

When you end a connection at another connection, PowerLogic places a tie dot at the junction. The resulting net name is the name of the added connection, not the existing connection, unless you assign a name to the existing connection.

To assign a fixed net name to a connection, you must first add the connection and then select a segment that is directly attached to a part type pin. Then, you can invoke Query mode and enter the new net name in the Query/Modify Net Name dialog box.

Optionally, you can add multiple copies of the connection after the initial connection location is indicated. See the "Step and Repeat" topic in this chapter for more information.

Adding Off-page References

To connect two pins on separate sheets of the schematic or at opposite ends of the same sheet, without a graphical connection, use an off-page reference with identical net names.

To add an off-page reference:

1. Specify the connection as described in the previous section.
2. Choose Offpage from the pop-up menu. The symbol attaches to the connection and follows the cursor movement.

   The default decal with the right pin is a source pin; the alternate decal with the left pin is a load pin. This pin type orientation plays a role when you create a hierarchical symbol of the sheet. All off-page sources are on the left of the hierarchical symbol outline, and all loads are on the right. You can place the default or choose Alternate from the pop-up menu.

   Optionally, you can rotate or mirror the symbol by choosing the appropriate command from the pop-up menu.
3. Locate the cursor and double-click to end the connection or choose Complete from the pop-up menu.

4. Type the net name in the dialog box and choose OK.

The sheet number for the connecting symbol appears next to the off-page symbol if the Show Off-Page Reference option is on in the Design tab of the Preferences dialog box.

Note: If you display the off-sheet numbers, you can line them up or stack them by specifying Numbers per Line in the Design tab.

Adding Power and Ground Connections

When you connect a part to ground or power nets, the normal convention is to use a special symbol to represent the net. The ground symbol ties the connection to ground. The power symbol adds voltage to connections.

1. To add a power or ground symbol:
2. Specify the connection as described in the "Adding Connections" topic in this chapter.
3. Choose Ground or Power from the pop-up menu. The symbol attaches to the connection and follows the cursor movement.

   Optionally, you can rotate or mirror the symbol by choosing the appropriate command from the pop-up menu. You can also modify the symbol representation by choosing Alternate from the pop-up menu. Choose Display PG Name from the pop-up menu to toggle the visibility of the net name for the power or ground symbol.
4. Locate the cursor and double-click to end the connection or choose Complete from the pop-up menu.

You can create and store a number of ground symbols in the library, each with its own net name. To select the appropriate ground symbol, choose Alternate from the pop-up menu. Three symbols are available with the standard library. You can modify these or add additional symbols.

PowerLogic assigns the net name associated with the ground symbol to the connection; for example, GND, Chassisground, or Analog-ground.

For each alternate power symbol, there is a different signal name: +5V, +24V, etc. To select the appropriate signal name, choose Alternate while placing the symbol.

PowerLogic displays the net name associated with the power or ground symbol in the status window. Use the Status Window to view the alternates and locate the net name that you want to tie the connection to.

Naming a Connection

When you add a connection, PowerLogic automatically assigns a name with the format $$$*nnnn*, where *nnnn* is a random number. You can specify a different net name to replace the assigned name. To modify the net name for a connection:

1. Choose the Query/Modify icon from the toolbar.
2. Select a terminating connection segment.
3. Type a new net name in the text box.

To place a bar over the net name, precede the text with the \ character. To place a bar over only a portion of text, enclose that section with the \ character. For example, typing MAIN_\CLK\ places a bar over CLK.

Note: The \ character factors into the 47 character limit for the net name.

Changing a Connection

Use Move mode to change a connection from one end point to another, without deleting the rest of the connection.

1. Choose the Move Mode icon from the toolbar or Set Command Mode/Move Mode in the pop-up menu.
2. Select the connection at any point on the segment entering or exiting the part or where it enters another connection. The connection follows the cursor movement. You can choose Angle from the pop-up menu to specify diagonal segments in the connection.
3. Indicate corners as needed or choose Add Corner from the pop-up menu. You can choose Del Corner from the pop-up menu to back up to the previous corner if necessary.
4. A connection cannot end in open space. Locate the cursor and double-click to end the connection or choose Complete from the pop-up menu.

To end a connection with a power, ground, or off-page reference symbol, choose the appropriate symbol from the pop-up menu during the last step of this procedure.

Hierarchical Design

PowerLogic supports top-down and bottom-up design. It is possible to create a schematic, then define the hierarchical symbol for the schematic, or to define a hierarchical symbol and then the underlying logic. You can also assign or unassign in the Query Hierarchical Component dialog box.

Level 0 is the top design level that contains the design contents. Numbered sheets from sheet 1 to 1024 represent the design contents. A PowerLogic design cannot exceed a total number of 1024 hierarchical and numbered sheets.

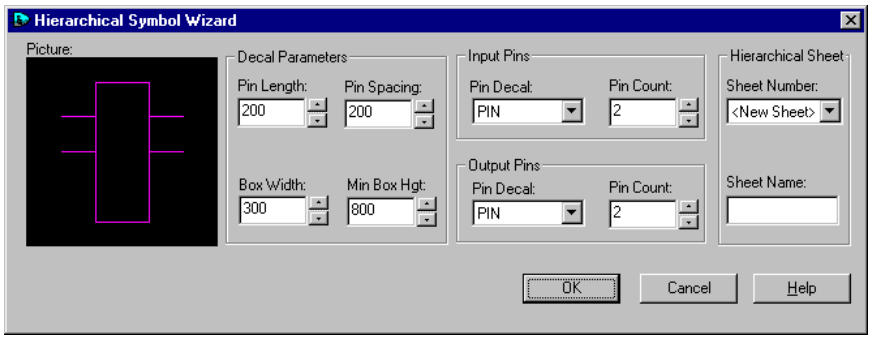
You can navigate the hierarchy with the View/Push and View/Pop commands.

When a hierarchical symbol is copied, all of the underlying logic represented by the symbol is also copied. PowerLogic updates the reference designator names of the underlying symbols. There is a one-to-one correspondence between hierarchy symbols and a schematic sheet. The underlying logic for duplicated hierarchical symbols is stored as a unique sheet, and can be edited without affecting the logic of other hierarchical symbols.

Creating a Hierarchical Symbol

You can create a new hierarchical symbol in either a top-down or bottom-up design structure. The symbol creation methodology varies depending on the structure you use. Choose the New Hierarchical Symbol icon from the Design toolbox to start the process. The Hierarchical Symbol Wizard appears.

Follow the instructions for Creating a Top-down Hierarchy or Creating a Bottom-up Hierarchy based on the requirements of your design or your architectural preference.



Picture

Displays the symbol outline before the symbol appears in the Part Editor.

Decal Parameters

Pin Length	Sets the distance from the terminal connection point and the decal outline. Type a value or use the arrows to indicate the pin length. Note: This option does not adjust the length of the pin decal.
Pin Spacing	Sets the distance between pins. Type a value or use the arrows to specify the distance.
Box Width	Sets the width of the decal outline. Pin decals are moved left or right to accommodate the box width. Type a value or use the arrows to specify the width.
Min Box Hgt	Sets the minimum height of the decal outline. If you enter a value larger than needed to accommodate the number of input or output pins, space is added to the bottom of the decal.

Pin Decal

Indicates the type of pin decal for input and output pins. A drop-down list box displays the available choices.

Pin Count

Specifies the number of input and output pins for a hierarchical symbol of a new sheet during top-down design.

Hierarchical Sheet

Lists the sheets in the set and a new sheet option for top-down design.

Sheet Name

Specifies a hierarchical symbol name within a maximum of 37 characters. You can use the first 32 characters for the textual portion of the name, the remaining five are reserved for the numeric portion.

Creating a Top-down Hierarchy

In top-down design, you create a hierarchical symbol sheet without an underlying logic definition.

Pin names tie together signals between hierarchical levels. PowerLogic ties together all instances of a common signal name, regardless of where they are found in the design, into a single connection net. You establish connectivity across the hierarchy by using the same signal name in the underlying logic.

If you use a top-down approach for hierarchy, you must ensure that the off-page signals in the underlying schematic are consistent with the pin names in the hierarchical symbol.

1. Choose the Design Toolbox icon from the toolbar. See the "Design Toolbox" topic in this chapter for more information.

2. Choose the New Hierarchical Symbol icon from the Design toolbox. The Hierarchical Symbol Wizard dialog box appears.
3. Choose New Sheet in the Sheet Number drop-down list box.
4. Type the symbol name in the Sheet Name text box.
5. Choose the Pin Decal for both input and output pins from the appropriate drop-down list box.
6. Specify the Pin Count for both input and output pins. Once you specify the pin details, you can optionally choose Preview to display the symbol outline.

7. Choose OK. The hierarchical symbol appears in the Part Editor window.

   Because the symbol is a hierarchical one, it has no pin numbers. The pin name associated with the hierarchical symbol pins identifies the net name of the connection that is tied to the pin. All pins on a hierarchical symbol must have a pin name. You cannot complete the hierarchical symbol until you assign a name to each pin.

8. Name pins in the Part Editor and choose File/Complete. This closes the Part Editor and the new symbol follows the cursor movement.

9. Move the symbol and indicate its location.

You can navigate this hierarchy with the View/Push and View/Pop commands.

Creating a Bottom-Up Hierarchy

In bottom-up design, you create a hierarchical symbol for an existing sheet that represents the underlying logic.

PowerLogic adds an input or output pin in the symbol for each off-page reference on the schematic sheet. PowerLogic places off-page references with a pin type of Source on the left side of the hierarchical symbol and off-page references with a pin type of Load on the right side.

Note: Pin types for off-page references are defined in the library under electrical.

1. Choose the File/Open command to load the schematic job file.
2. Switch to the sheet that you want to add the hierarchical symbol to, or add a new schematic sheet to the sheet set with the Setup/Sheets command.

3. Choose the Design Toolbox icon from the toolbar. See the "Design Toolbox" topic in this chapter for more information.

4. Choose the New Hierarchical Symbol icon from the Design toolbox. The Hierarchical Symbol Wizard dialog box appears.
5. Choose an existing sheet from the Sheet Number drop-down list box.
6. Type the symbol name in the Sheet Name text box.
7. Choose the Pin Decal for both input and output pins from the appropriate drop-down list box. Once you have specified the pin details, you can optionally choose Preview to display the symbol outline. The number of input/output pins is grayed because PowerLogic knows the number of pins of each pin type on an existing sheet.
8. Choose OK. The hierarchical symbol appears in the Part Editor window. An input or output pin is added in the symbol for each off-page reference.
9. Choose File/Complete. This closes the Part Editor and the new symbol follows the cursor movement.
10. Move the symbol and indicate its location.

You can navigate this hierarchy with the View/Push and View/Pop commands.

Modifying a Hierarchical Symbol

To modify a hierarchical symbol:

1. Choose the Edit Part/Edit Hierarchical Symbol icon from the Design toolbox.
2. Select the hierarchical symbol.
3. Modify the symbol in the Part Editor.
4. Choose File/Complete when you finish the changes.

If you add terminals to a hierarchical symbol, ensure that you give them pin names, and that the names also appear as off-page references in the sheet with the underlying schematic.

Pushing Into the Hierarchy

Use the View/Push Hierarchy command to look inside or push down into a hierarchical symbol to view the underlying logic.

To navigate into a hierarchical symbol:

1. Choose the View/Push Hierarchy command from the menu.
2. Select the hierarchical symbol.

If there is no underlying schematic representation of the selected symbol, a blank sheet appears.

Moving Up the Hierarchy

Use the View/Pop Hierarchy command, while in a hierarchical sub-schematic, to replace the current sheet with its corresponding hierarchical symbol.

Swapping Reference Designators

To swap two reference designators, choose the Swap Reference Designator icon from the Design toolbox.

1. From the Design toolbox, choose the Swap Ref. Des. Icon.
2. Select the first part.
3. Select another part to swap its reference designator with.

   If the reference designator prefixes of the selected parts are different, this swap also updates the reference designators of other parts in the package.

Swapping Pins

To swap two part pins, choose the Swap Pins icon from the Design toolbox. PowerLogic swaps the pin number, name, and terminal of the pins you select. You can also reverse pins when you place a part with the Mirror X and Mirror Y commands on the pop-up menu. This is often useful for improving the flow of the schematic.

1. Choose the Swap Pins icon from the Design toolbox.
2. Select the first pin number.
3. Select the pin number to swap with.

   The *Keep connections tied to same electrical pin?* prompt appears. Choose Yes to preserve the connections with the pin numbers. Choose No to reconnect to the renumbered pins.

Bus Toolbox

Use the Bus toolbox to define bus structures in PowerLogic. A bus is a collection of related connections that are grouped together into a single graphical symbol (a special thickened line). You can add, name, modify, connect, or delete busses as you would connections.

To make the Bus toolbox visible, use the Window/Bus Toolbox command, or choose the Busses icon from the toolbar.

	Add Bus	Adds and names a new bus. See the "Adding a Bus" topic in this chapter for more information.
	Split Bus	Bisects a selected segment with a new, perpendicular segment. See the "Splitting a Bus" topic in this chapter for more information.
	Extend Bus	Continues the bus definition from either end point. See the "Extending a Bus" topic in this chapter for more information.
	Move Bus Segment	Repositions the selected bus segment. See the "Moving a Bus Segment" topic in this chapter for more information.
	Delete Bus Segment	Removes the selected bus segment. See the "Deleting a Bus Segment" topic in this chapter for more information.

Adding a Bus

To add a bus:

1. Choose the Busses icon from the toolbar.
2. Choose the Add Bus icon from the Bus toolbox or Add Item/Add Bus from the pop-up menu.
3. Indicate the starting point. The bus follows the cursor movement.
4. Indicate bus corners as needed or choose Add Corner from the pop-up menu. You can choose Del Corner from the pop-up menu to back up to the previous corner if necessary.
5. Double-click to end the bus in open space or choose Complete from the pop-up menu.
6. Type the bus name in the text box or choose an existing name from the drop-down list box, and then choose OK. See the "Naming the Bus" topic in this chapter for more information. The name follows the cursor movement.

   Optionally, you can choose Rotate from the pop-up menu before you place the text. Indicate the name location.

Naming the Bus

The bus name consists of two parts: the base name and the bit range of the bus. The format is *BASE*[*nn:mm*] where *nn* is the lowest bit number and *mm* is the highest bit number, for example PADS[00:15].

The sheet number also displays with the bus name if the Show Off-Page Reference option is on in the Design tab of the Preferences dialog box.

The total bus name, including the base name, brackets, and colon, cannot exceed 47 characters. The bit numbers must be in the 00 to 99 range and cannot contain alphabetic characters.

PowerLogic defines a bus that appears on the same sheet, or any other sheet, with an identical name as the same bus.

Adding Connections to a Bus

When connecting to a bus, the connection must be perpendicular to the bus. PowerLogic adds an angled bus-tap segment at the point of connection and gives the connection the next sequential net name in the bus.

Confirm or type the net name in the dialog box. The text follows the cursor movement. Position the text and indicate its location. Optionally, you can swap the bus tap angle by choosing Swap Angle from the pop-up menu.

You can add multiple copies of the connection after the initial connection location is indicated. See the "Step and Repeat" topic in this chapter for more information.

Splitting a Bus

To create new corners or a 90 degree turn in the bus:

1. Choose the Split Bus icon from the Bus toolbox or Modify Bus/Split Bus in the pop-up menu.
2. Select the bus segment. The new and divided segments follow the cursor movement.
3. Move the cursor to define the segment length. You can choose Swap Corner from the pop-up menu to define the end point in the opposite direction.
4. Indicate the end point of the new segment to complete the split.

Extending a Bus

To extend a bus segment:

1. Choose the Extend Bus icon from the Bus toolbox or Modify Bus/Extend Bus in the pop-up menu.
2. Select either end point of the bus. The end segment follows the cursor movement.
3. Indicate corners as needed for the extension or choose Add Corner from the pop-up menu.
4. Indicate the new end point or choose Complete from the pop-up menu.

You cannot use the Extend Bus command to shorten a bus segment. To shorten a bus, delete the long segment, then use Extend Bus to redefine that segment. Alternately, you could use the Group command to shorten the bus segment.

Moving a Bus Segment

To reposition a bus segment:

1. Choose the Move Bus Segment icon from the Bus toolbox or Modify Bus/Move Bus Segment in the pop-up menu.
2. Select the bus segment. The segment follows the cursor movement.
3. Indicate the new location.

Deleting a Bus Segment

To remove a segment of the bus:

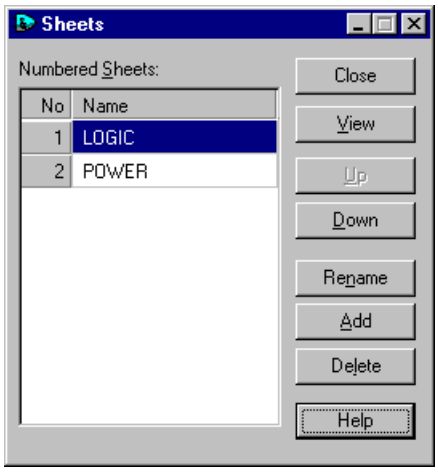
1. Choose the Busses icon from the toolbar.
2. Choose the Delete Bus Segment icon from the Bus toolbox or Modify Bus/Delete Bus Segment in the pop-up menu.
3. Select the bus segment.

If the deletion splits the bus into two separate busses, the bus name appears, identifying any unnamed bus segments. Each bus retains the same properties.

Sheets

Use the Sheets dialog box to edit the sheet set of the current schematic in the work area. Choose the Setup/Sheets command to open this dialog box. Using Sheets allows you to add and delete sheets from the set and to modify sheet names and the numeric order of the set. You can create up to 1024 sheets.

Double-click on a sheet name to view it in the work area. Choose a sheet in the list box, pause, and then choose it again to rename the sheet. Spaces are not valid characters in a sheet name. You can also use the View and Rename buttons for these functions.



Numbered Sheets

Itemizes the sheets of a design in a list box.

View

Displays the selected sheet in the work area.

Up

Renumbers the selected sheet with the value n-1.

Down

Renumbers the selected sheet with the value n+1.

Rename

Creates or modifies the name assigned to a sheet.

Add

Adds a new sheet to the list box and the design and displays the new sheet.

Delete

Deletes the selected sheet from the design.

Modes of Operation

PowerLogic has three basic modes of operation: Group Select mode, Object Select mode, and Verb Mode. In addition to these operational modes, PowerLogic has Zoom mode. Zoom mode overrides all other modes until you specify another mode.

Group Select Mode

Group Select mode is the default mode of operation when you start PowerLogic. You perform group operations in Group Select mode. PowerLogic is in Group Select mode when the Select Arrow on the Standard toolbar is selected and when all filter controls are turned off on the Object Selector toolbar. Selecting the Group Selector icon on the Object Selector toolbar can turn off the filter controls. See the "Group Operations" topic in this chapter for more information.

Verb Mode

Commands that operate on a selected object are considered *verb* commands. In Verb mode you select the mode, then select the objects on which you want to use the verb.

Place PowerLogic in Verb Mode by choosing one of the following commands from the Standard toolbar, then select an object on which to perform the command. When you move the cursor off the toolbox, a small V appears on the cursor to show that PowerLogic is in Verb mode.

Copy

Move

Delete

Query/Modify

Copy Mode

Use Copy mode to replicate existing parts, text, and drafting objects. When you copy parts, PowerLogic automatically increments the reference designator and gate modifier.

To copy an object:

1. Choose the Copy Mode icon from the toolbar or Set Command Mode/Copy Mode in the pop-up menu.
2. Select the item to copy. A duplicate of the item follows the cursor movement.

   Optionally, and depending on the available options for the selected item, you can adjust the orientation and other settings from the pop-up menu before placement. Position the item and indicate its location to complete the copy.
3. When you finish placing copies of this item, choose Cancel from the pop-up menu.

PowerLogic remains in Copy mode until you choose another mode, toolbox, or the Select icon from the toolbar.

Copying a Hierarchical Symbol

When you copy a hierarchical symbol, the sheets that the symbol references will also be copied and added as new sheets to the schematic. PowerLogic assigns new reference designators to the part types in the copied sheets. If the referenced sheet also contains hierarchical symbols, then PowerLogic also copies the sheet(s) referenced by those symbols, and so on, down the entire sub-tree of the hierarchy.

When you select a hierarchical symbol in copy mode, the *Copy hierarchical component. Are you sure (Y/N)?* prompt appears. If you choose Yes, PowerLogic copies the hierarchical symbol, and all directly or indirectly referenced sheets. If any numbered sheets are referenced, PowerLogic assigns the copies new sheet numbers.

Step and Repeat

Use Step and Repeat to multiply objects as you place them during an add or copy operation. Step and Repeat is available in the Schematic Editor and the Decal Editor. In the Schematic Editor the Step and Repeat command copies parts, connections, text, or drafting items. In the Decal Editor the Step and Repeat command copies terminals, text, or drafting items.

To use Step and Repeat:

1. Select an object during an add or copy operation. While a dynamic object is attached to the cursor, choose Step and Repeat from the pop-up menu. The Step and Repeat dialog box appears.

Note: When adding a new object in the Schematic Editor, you must place the first object manually before you can use Step and Repeat.

---

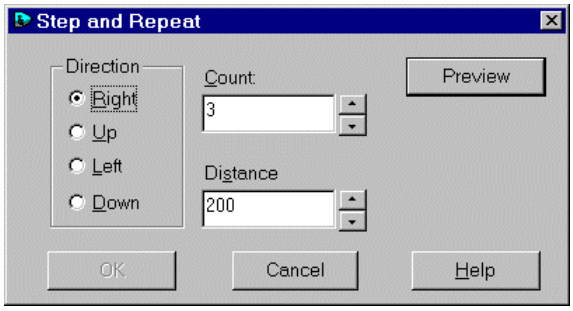
2. Set the options for the array. Options include the direction of placement, the number of objects to place, and the distance between objects.

3. Choose the Preview button to view the placement of the multiple objects based on the options you set. The placement of the objects is based on the location of the original object selected. You can not place objects until you preview the placement.

Note: Zoom Mode is available during Step and Repeat.

4. Once the placement preview is correct, choose OK to place the objects.

After using Step and Repeat, the original dynamic object is still attached to the cursor. You can continue to add or copy using this object, or you can press Esc to end the operation.



Direction	Defines the direction of the replication in the array. For example, if you choose Right, the copies flow directly to the right of the original.
Count	Sets the number of replications in the array. Type the number of copies you want to place in the text box, or click on the arrow buttons.
Distance	Sets the distance between each copy of the object in the array; based on the current Snap to Grid preference settings. Type the distance value in the text box, or click on the arrow buttons.
Preview	When pressed, displays a preview of the array. The OK button is not available until you preview the Step and Repeat operation.

Copying Connections

When copying connections, only connections with valid start and end points remain after the Step and Repeat operation, even if invalid connections appear during Preview. A copied connection can start on a component pin or on another connection segment. A copied connection can end on a component pin, a connection segment, or a bus segment.

PowerLogic automatically assigns new bus netnames to bus segment copies. The new netname is based on the original bus segment netname, plus one. For example, if you make a copy of bus segment D00, the copy is assigned the netname D01.

Semiautomatic Step and Repeat

If you manually place a copy within 0.5" of the original (in both X and Y directions), the next connection is placed at the same offset. Then press the Space Bar to place a new copy. You can continue to use this semi-automated Step and Repeat process to place subsequent copies.

Move Mode

Use Move mode to adjust the position of existing parts, connections, text, and drafting objects.

To move an object:

1. Choose the Move Mode icon from the toolbar or Set Command Mode/Move Mode in the pop-up menu.
2. Select the object to move. The object follows the cursor movement.

   Optionally, and depending on the available options for the selected object, you can adjust the orientation and other settings from the pop-up menu before placement. Position the object and indicate its location to complete the move.

PowerLogic remains in Move mode until you choose another mode, toolbox, or the Select icon from the toolbar.

Moving Text

When you add a part to the schematic, all of the text associated with the symbol (reference designator, part name, pin number, etc.) appears in a predefined location. You can move the text to make room for other parts or connections or to make the schematic more readable.

You can redefine the characteristics of the selected text as you relocate the string, when PowerLogic resumes the Adding Text function in Move mode.

You can move a pin number within a bounding area of one inch from its pin terminal.

You can move the off-page sheet numbers line(s) independently of the net name.

Moving Connections

When you move a part, existing part connections follow the part movement. After part repositioning, you might need to move connections to improve the flow.

You can reposition a connection segment that is not a connection end point or you can reconnect a segment end point. See the "Changing a Connection" topic in this chapter for information on disconnecting from a part or connection and reconnecting to another part or connection.

Splitting a Connection

To create new corners or a 90-degree turn in a connection:

1. Choose the Move Mode icon from the toolbar or Set Command Mode/Move Mode in the pop-up menu.
2. Select the segment.
3. Choose Split Segment from the pop-up menu. The new and divided segments follow the cursor movement.
4. Move the cursor to define the segment length. You can choose Swap Corner from the pop-up menu to define the end point in the opposite direction.
5. Indicate the end point of the new segment to complete the split.

Delete Mode

Use Delete mode to completely remove parts, connections, unconnected busses, text, drafting objects, and hierarchical symbols from the design. If you delete a part with connections, PowerLogic also deletes the connections.

1. Choose the Delete Mode icon from the toolbar or Set Verb Mode/Delete Mode in the pop-up menu.
2. Select the object to delete.

PowerLogic remains in delete mode until you choose another mode, toolbox, or the Select icon from the toolbar.

Deleting a Hierarchical Symbol

When you select a hierarchical symbol in delete mode, the message *OK to delete sheets associated with the hierarchy - Warning: there is no Undo for this operation* appears. If you choose Yes, another prompt *Delete associated sheet (Y/N)?* appears. If you choose Yes, PowerLogic deletes the hierarchical symbol and any associated sheets throughout the hierarchy. If you choose No, and it is a hierarchical symbol without a sheet number, PowerLogic assigns it the next unused sheet number and it becomes a normal, numbered sheet.

Object Select Mode

In Object Select mode you select the object, then select a verb with which to perform an action. Select an object using the Object Selector toolbar, or pop-up menu, then choose a verb from the Standard toolbar or the pop-up menu.

The type and quantity of objects selected determines which Verb modes are available:

- When a single Gate is selected, all Verb modes are available.
- When more than one Gate or Part is selected, Delete mode only is available.
- When more than one Gate of a Multigate Part is selected, Copy and Move modes are not available.
- When a Net(s) or Pin(s), or a combination of different objects (like Parts and Pins) is selected, no Verb modes are available.

Group Operations

A group is a collection of parts, connections, drafting objects, and free text. Once a group is created, it can be moved, copied, deleted, or saved to file for use in another design.

Before creating a group, you must place PowerLogic in Group Select mode, using one of three methods:

- Choose the Group icon from the Object Selector toolbar.
- Choose Edit/Filter/Select Group.
- With no objects selected, choose Select Group from the pop-up menu.

To define a new group:

1. Enter Group Select mode.
2. Use the left mouse button to drag a rectangle around the objects that define the group. The included objects are highlighted.

Before using or saving a group, you can enhance it as described in the following sections.

Setting a New Origin for the Group

To set a new origin, or insertion point for the group:

1. Click the right mouse button to display the pop-up menu.
2. Choose Set Origin. A bullseye attaches to the cursor.
3. Move the cursor and select a new origin with the left mouse button.

Removing Connections Outside the Boundary

By default, connections that extend outside the group are clipped so they stop at the rectangle used to define the group. To exclude the entire connection from the group:

1. Click the right mouse button.
2. Turn the Cut Connections at Edge option off.

Adding to or Removing from a Group

When creating a new group, there may be objects inside the group definition you want to exclude from the group or objects outside of the group that you want to include. Use the Ctrl key and the left mouse button to add or remove objects from the group.

To remove objects:

1. Press the Ctrl key.
2. Choose the objects to remove. They are not highlighted when removed.

To add objects:

1. Press the Ctrl key.
2. Choose the objects to add. They are highlighted when added.

Note: You must choose the reference designator text associated with the part when choosing a part to add or remove from the group.

You cannot remove connections tied to highlighted parts in the group without removing the parts. Use the Cut Connector at Edge option to remove connections that extend outside the group boundary.

Moving a Group

You can move a defined group to any location on the sheet with the Move command. See the "Copying a Group" and "Pasting a Group" topics in this chapter to insert a group on a different sheet. After defining a group:

1. Choose Move from the pop-up menu or the Edit menu. The outline of the group is attached to the cursor.
2. Move the cursor to the location you want and click the left mouse button to anchor the group.

You can place a temporary copy of a group to check for positioning, connection arrangement, etc.:

1. Place the cursor at the location you want.
2. Choose Draw Group from the pop-up menu. The contents of the group appear, but the group remains attached to the cursor.
3. Use Erase Group from the pop-up menu to erase the temporary copy placed with Draw Group.

Copying a Group

You can copy all of the items in a group. When a group is copied, the names of the parts are automatically incremented and assigned. Reference designator names are automatically assigned using the next available names, and gaps in the reference designator name sequence are filled.

Note: If the group contains hierarchical symbols, the message *Group contains Hierarchy - Warning: there in no Undo for this operation* appears. Choose OK to continue the operation or Cancel to abort.

After defining a group:

1. Choose Group Copy from the pop-up menu or the Edit menu. The outline of the group is attached to the cursor.
2. Move the cursor to the desired location and click the left mouse button to anchor the copy.

Repeat these steps to create additional copies.

You can set PowerLogic to display only the group's outline during copy operations. See the "Displaying Groups During Move or Copy" topic in this chapter for more information.

Displaying Groups During Move or Copy

When moving or copying a group, in addition to displaying the group outline, PowerLogic continually redraws and displays the outline of each part and connection in the group. Use Erase Group and Draw Group to control how group information is displayed.

Erase Group

Use Erase Group to display only the group's outline. This increases performance when moving or copying larger group. In addition, not having all connection lines attached to the cursor makes target areas easier to locate.

To display only the group outline:

While moving a group, choose Erase Group from the pop-up menu. The outline of the group attaches to the cursor at the group's origin.

Draw Group

Use Draw Group to return to normal display; dynamically attach and display all connections and part outlines in the group during move or copy operations:

While moving a group, choose Draw Group from the pop-up menu.

Note: Display settings established with Erase Group and Draw Group remain in affect until you change them.

Deleting a Group

When deleting a group, the portions of connections in the group that are outside the boundary of the group are retained, with an automatically assigned off-page symbol. Unnamed connections will be named uniquely with a net name of the form ###xxxx.

After defining a group, choose Delete from the pop-up menu or the Edit menu.

Note: If the group contains hierarchical symbols, the message *Group contains Hierarchy - Warning: there in no Undo for this operation* appears. Choose OK to continue the operation or Cancel to abort.

Copying a Group to a File

Use Copy to File to save the contents of a group for use on other schematic sheets or a different design. Saved groups are stored in the \padspwr\logicfiles directory with a .grp extension. After defining a group:

1. Choose Copy to File from the pop-up menu or the Edit menu. The Group Save File dialog box appears.
2. Type a name in the File Name area, or use the default.
3. Choose Save.

Pasting a Group from a File

Use Paste from File to insert a group saved with the Copy to File command. When the group is pasted into a sheet, all of the parts are named with the next available reference designator names. Use the Preserve Reference Designators on Paste option in the Design tab to control renaming of reference designators. See the "Preserving Reference Designators" topic in this chapter for more information.

Note: If the group contains hierarchical symbols, the message *Group contains Hierarchy - Warning: there in no Undo for this operation* appears. Choose OK to continue the operation or Cancel to abort.

Nets with defined names are retained. Unnamed nets are given default signal names.

To paste a group into a design:

1. Choose Edit/Paste From File. The Load Group File dialog box appears.
2. Choose the group from the list.
3. Choose Open. The group is placed at the current cursor location. If the design contains parts with the same reference designation as those in the group, a report is generated. See the "Preserving Reference Designators" topic in this chapter for more information.
4. The Group is attached to the cursor. Select the desired location.

Preserving Reference Designators

When you paste a group in a design, PowerLogic uses the reference designators current in the saved group file. If conflicts with current parts exist, PowerLogic renames the duplicate reference designators and generates an error report containing the renamed parts. The report displays in the default text editor.

To restart numbering of the reference designators or to use available reference designator numbers, turn off the Preserve Reference Designators on Paste option in the Design Tab. If you turn this option off and paste a group into a new design, reference designation starts at the first number. For example, U1.

Cut, Copy, and Paste

Use Edit/Cut to remove a group and place a copy in a temporary .grp file. Connections that extend outside the group have an off-page reference symbol added at the break point.

Use Edit/Copy to remove a group and place a copy into the Windows paste buffer. Connections that extend outside the group have an off-page reference symbol added at the break point.

Once in the buffer, you can use Edit/Paste to place the copy anywhere on the design, or you can close the design and open a different design to paste into. You can paste the same copy repeatedly; the copy remains in the temporary file until it is overwritten by a new copy action.

Copy as Bitmap

The Copy as Bitmap command lets you define a rectangular area to copy graphics information to the temporary .grp file as a bitmap image. Once copied, you can, for example switch to Microsoft Word and use the Edit/Paste command to insert the bitmap into a document.

To create a bitmap:

1. Area-select the area to copy.
2. Choose Edit/Copy as Bitmap.

All items in the rectangle including the background, dot grid, color, etc. are copied. Open the application in which to place the bitmap and use the Paste command to place the image.

Note: In addition to creating a bitmap of a defined area, you can copy the entire work area as a picture to insert using the Paste Special command in products that support this feature.

Query/Modify

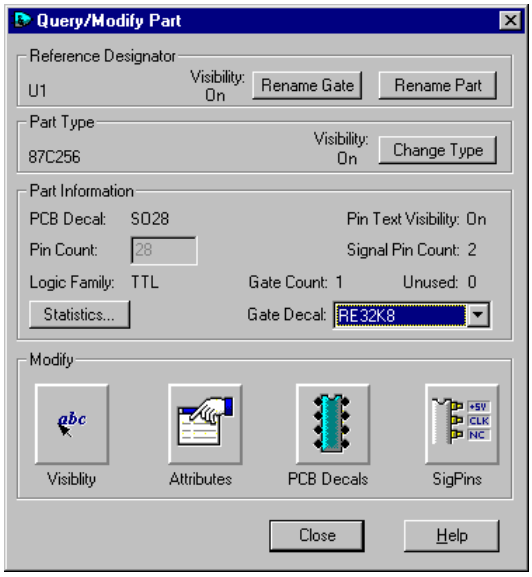
Choose the Query/Modify icon to place PowerLogic in Query/Modify mode. Use this mode to get information on and modify objects in PowerLogic. When you select an object in Query/Modify mode, a dialog box or information window appears.

Query/Modify Part

Use the Query/Modify Part dialog box to create and edit part options. You can also define signal pins and control the visibility of options assigned to the part.

1. Choose the Query/Modify icon from the toolbar.
2. Select the part to modify by selecting a construction line or text string associated with the part. The Query/Modify Part dialog box appears.
3. Change the settings, then choose Close.

This dialog box contains several other dialog boxes. Before you can use any of these other dialog boxes, apply changes you make in the Query/Modify Part dialog box to the design.



Rename Gate

Changes the reference designator of the selected gate. You cannot assign reference designators already in use or an unused gate of a part with a different part type.

1. Choose Rename Gate from the Query/Modify Part dialog box. An information window appears.
2. Type the new gate reference designator information.
3. Choose OK.

Rename Part

Changes the reference designator for the selected part or gate. All gates are renamed if you change the reference designator of one gate of a multigate part. You cannot assign reference designators already in use.

1. Choose Rename Part from the Query/Modify Part dialog box. An information window appears.
2. Type the new part reference designator information.
3. Choose OK.

Change Part Type

Changes the selected part to a new part type. The new part type can already exist in the schematic or in the parts library. See the "Change Part Type Dialog Box" topic in this chapter for more information.

1. Choose Change Type from the Query/Modify Part dialog box. The Change Part Type dialog box appears.
2. Choose a new part type.
3. Choose OK.

Note: You can update the part definition in the schematic with a modified version in the library. Choose the same part name, and then choose All Parts This Type in the Apply Update to area.

Pin Count

Use the Pin Count text box to control the upper limit of pin numbers available when assigning signal pins to parts without a PCB Decal.

If there is a PCB Decal assigned to the part, this control shows the decal pin count and the edit box grays, since the decal definition determines the number of pins.

1. Choose the Query/Modify icon from the toolbar.
2. Select the part to modify by selecting a construction line or text string associated with the part.
3. Type the new pin count.

Statistics

Use Statistics to display gate and connection information for the selected part. This information is displayed in the default text editor so you can save the contents to a file.

Gate Decal

Use the Gate decal drop-down list box to change the gate decal of the selected gate or part to one of the predefined alternate decals.

Part Text Visibility

Controls the display of text associated with the selected part. Use the Visibility button to control the visibility of one part or all parts of the same type. See the "Query/Modify Part Text Visibility" topic in the "Working With Attributes" chapter for more information.

Part Attributes

Assigns or modifies part attributes, which is information about the part such as manufacturer and cost. Use the Attributes button to modify this information. See the "Query/Modify Part Attributes" topic in the "Working With Attributes" chapter for more information.

PCB Decals Assignment

Assigns alternate PCB decals to schematic parts. See the "Query/Modify PCB Decals Assignment" topic in this chapter for more information.

Part Signal Pins

The Signal Pins button assigns any unused pins as additional signal pins. When you create and store a part type in the library, the standard power and ground pins for part types are defined.

Signal pins assigned during part creation cannot be modified through this dialog box. Use the Assigning Signal Pin Names to Parts dialog box in the library manager to modify signal pins assigned during part construction.

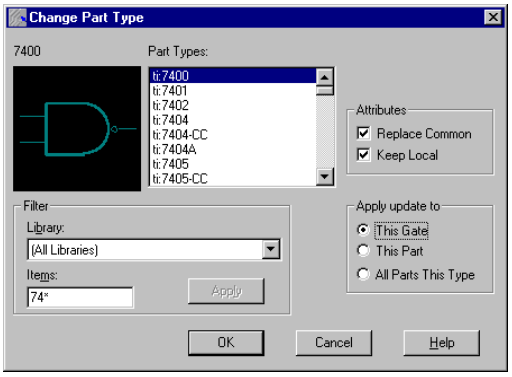
1. Choose the Query/Modify icon from the toolbar.
2. Select a part to add or to modify the signal pins of by selecting a construction line or text string associated with the part.

3. Choose the Signal Pins icon from the Query/Modify Part dialog box. The Query/Modify Part Signal Pins dialog box appears.
4. Choose an unused pin, then choose Add.

You can also assign a signal name and trace width in the Signal Pins area. Use the Apply Update area to apply the signal pin changes to the selected part or all parts of the same type.

Change Part Type Dialog Box

Use Change Type to change the selected part to a new part type, which can already exist in the schematic or in the library. Choose Change Type from the Query/Modify Part dialog box. The Change Part Type dialog box appears.



Part Types

Displays in the list box the part types that PowerLogic matched to the search filter settings. Choose the part you want to change from this list. The decal of a selected part displays in the area to the left of the list box.

Replace Common

Updates common attributes with the values contained in the new part.

Keep Local

Retains attributes in the current part even if they do not exist in the new part.

Filter Area

Searches the selected library for a specific part or item name or for names that match a wildcard text string. See the "Preface" for more information.

This Gate

Updates only the selected gate.

This Part

Updates only a part or all associated gates of a part.

All Parts of This Type

Updates all matching gates and/or parts in the design.

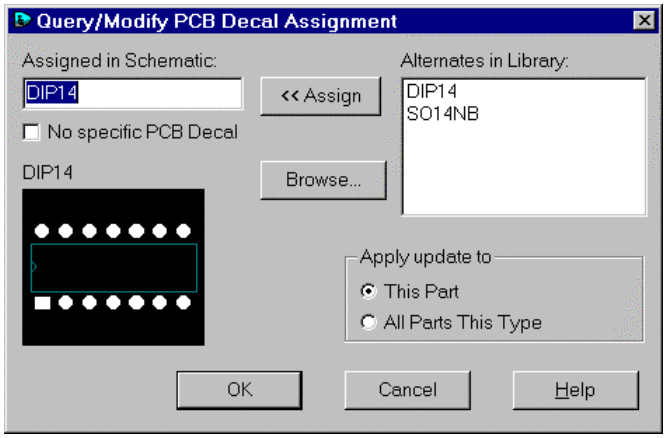
Query/Modify PCB Decal Assignment

Use the Query/Modify PCB Decal Assignment dialog box to assign alternate PCB decals to schematic parts. Decal names are included in the netlist file to display the proper decal, or footprint, when the file is imported into the PCB design file. You can choose a decal assigned as an alternate during part creation, override the decal with one from the library, or enter a name for an undefined decal you plan to create later in PowerPCB.

1. Choose the Query/Modify icon from the toolbar.
2. Select a part from the schematic by selecting a construction line or text string associated with the part. The Query/Modify Part dialog box appears.

3. Choose the PCB Decals icon. The Query/Modify PCB Decal Assignment dialog box appears.
4. Choose an Alternate Decal from the current part type definition in the library, or choose Browse to retrieve a new decal from any library.
   To enter a name for a decal you plan to create later in PowerPCB, type the new decal name in the Assigned in Schematic text box.
5. Choose OK to exit the dialog box.

All assigned decals must have the same number of pins.



Assigned in Schematic

Displays the name of the currently selected decal as it is assigned to the schematic from the current library. To enter a name for a decal you plan to create later in PowerPCB, type the new decal name in the Assigned in Schematic text box.

Assign Button

Assigns the decal selected from the Alternates in Library field to the part.

Alternates in Library

Provides a list of PCB decals from the current library for the part type of the selected part. Use Browse to search for other PCB decals.

No Specific PCB Decal

When on, the default decal assigned to the part type is used when the netlist is imported to PowerPCB; no decal assignment appears in the netlist.

Preview Area

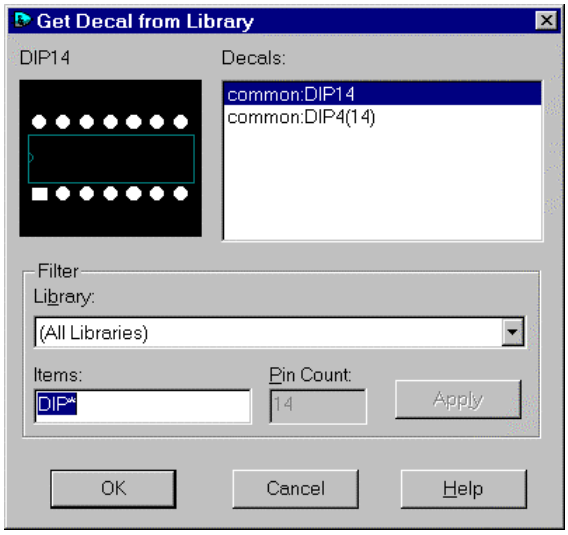
Displays the currently selected decal.

Browse Button

Opens the Get Decal from Library dialog box.

Get Decal from Library Dialog Box

When you choose the Browse button from the PCB Decal Assignment dialog box, the Get Decal from Library dialog box appears.



Decals

Provides a list of PCB decals from the library defined in the Filter area.

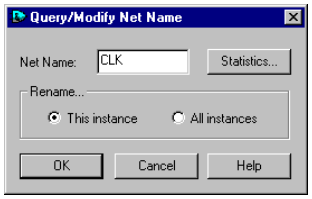
Filter Area

Searches the selected library (or libraries) for specific decal names that match a wildcard text string. See the "Preface" for more information.

Query/Modify Net Name

Use Query/Modify Net Name to change the system created net name.

1. Choose the Query/Modify icon from the toolbar.
2. Select a connection where it enters or exits a part. Selecting any other segment of the connection will not display this dialog box.
3. Enter a new name for the net in the Net Name text box or choose an existing name from the drop-down list box. If you choose a name from the drop-down list box, the message *Net <netname> already exists - OK to combine nets (Y/N)?* appears. Choose Yes to combine the nets. Choose No to enter a different net name.
4. Choose OK.



The new net name is placed at the end of the connection. Use Move Mode to adjust the placement of the net name text.

Use Statistics to display connection information in the default text editor.

Use the Rename area to apply the new net name to the selected connection or to all instances of this connection.

Note: Selecting This Instance will cause the net to be split into two separate nets.

Net Name

Displays the net name. You can also type a new net name. The new net name is placed at the end of the connection. Use Move mode to adjust the placement of the net name text.

Statistics

Displays connection information for the selected net in the default text editor.

This Instance

Applies the new net name to the selected connection.

Note: Selecting This Instance splits the net into two separate nets.

All Instances

Applies the new net name to all instances of this connection.

Query/Modify Bus Name

Use Query/Modify mode to change the name of a bus.

1. Choose the Query/Modify icon from the toolbar.
2. Select a bus name or bus segment.
3. Enter a new prefix name for the bus or choose a name from the drop-down list box.
4. Choose OK.

If specified, the new bit range nn:mm must be equal to or greater than the old bit range. The signals attached to the bus will keep their old bit number, but take the new name.

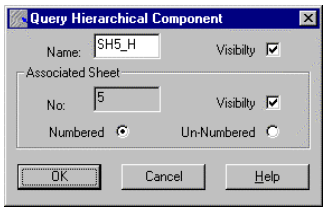
> Note: Renaming a bus name will split the selected bus and its associated net names from other busses with the same name. Use the 'S' modeless command to locate other bus names if they need to be updated. See the "Modeless Commands" topic in the "PowerLogic's Graphical User Interface" chapter for more information.

Query Hierarchical Component

When created and added to an existing sheet, you cannot access hierarchical components through the Sheet list box in the toolbar unless the parent sheet is displayed. They are also excluded from the Sheet command in the Setup menu.

Use the Query Hierarchical Component dialog box to assign the next available sheet number to a hierarchical component, making it accessible from the Sheet list box even when a sheet other than the parent sheet is displayed. This also allows you to use the Sheet command in the Setup menu to modify the sheet name or the numeric order.

1. Choose the Query/Modify icon from the toolbar.
2. Select the hierarchical component. The Query/Modify Hierarchical Component dialog box appears.
3. Modify the settings for sheet numbering and visibility.
4. Choose OK.



Name

Renames the hierarchical component.

Visibility (next to Name)

Displays the name on top of the hierarchical component in the schematic.

Visibility (above Unnumbered)

Displays the sheet number in the schematic.

Numbered

Assigns the hierarchical component the next available sheet number. The assigned sheet number appears in the field above this option.

Unnumbered

Removes a sheet number assignment from a hierarchical component.

Editing Off-Page Symbol Sheet Numbers Per Line

The displayed sheet number appearing next to an off-page reference allows, by default, 5 sheet numbers to be listed in a single text string, separated by commas.

Use Query/Modify mode to edit the number of off-page symbol sheet numbers per line. For example, if set to one, and there are two sheets containing the same net, the sheets will be listed in two lines.

1. Choose the Query/Modify icon from the toolbar.
2. Select the off-page reference text string. This displays an information dialog box.
3. Change the settings for sheet numbers per line.
4. Choose OK.

See the "Design Tab" topic in the "Design Setup and Editing" chapter for information on setting the default sheets per line and controlling the display of sheet numbers on off-page reference symbols.

Chapter 5
Working With Attributes

This chapter discusses using attributes in PowerLogic, including the different points at which you can create attributes and using attribute labels. The topics in this chapter include:

Attributes
Creating New Attributes in a Schematic
Managing Attributes in a Schematic
Using Attribute Labels

Attributes

Attributes allow you to associate information with a schematic symbol. Attributes are made of two parts, an attribute name and its corresponding value. For example, you can create an attribute named Part Description and assign it a value of Hex Inverter.

You can create attributes that get assigned to every part in the library, every part in the open design, a part type, or to only one part in the design.

- To add an attribute to all parts in a library, create the attribute in the Manage Library Attributes dialog box. See the "Creating Attributes in the Library" topic in the "Library Operations" chapter for more information.
- To add an attribute to, rename an attribute in, or delete an attribute from the library, alter the attribute in the Manage Library Attributes Dialog Box. See the "Manage Library Attributes Dialog Box" topic in the "Library Operations" chapter for more information.
- To add an attribute to every part in the open design, create the attribute in the Managing Schematic Attributes dialog box. See the "Creating Attributes in a Schematic" topic in this chapter for more information.
- Use the Browse Library Attributes dialog box to browse and list all attribute names from libraries specified in the Library List dialog box. See the "Browse Library Attributes Dialog Box" topic in the "Library Operations" chapter for more information.
- To add an attribute to a specific part, create the attribute using the Attributes tab while editing the part. See the "Attributes Tab" topic in the "Part Editor Operations" chapter for more information.
- To add an attribute to a part on the schematic without updating the part type, use the Query/Modify Part Attributes dialog box. See the "Query/Modify Part Attributes" topic in this chapter for more information.

- After creating attributes, use the visibility controls to display them in the schematic. See the "Query/Modify Part Text Visibility" topic in this chapter for more information.
- You can create placeholders in CAE Decals so attributes are placed in predefined locations. See the "Using Attribute Labels" topic in this chapter for more information.

When you add a part to a design using Add Part, attributes defined in the part are added to the list of available attributes. You can then use this attribute in other parts. See the "Managing Attributes in a Schematic" topic in this chapter for more information.

Creating New Attributes in a Schematic

To create a new attribute and automatically assign it to every part in a design:

1. Choose Tools/Attribute Manager. The Manage Schematic Attributes dialog box appears.
2. Choose Add Attr. The Add New Attribute dialog box appears.
3. Type a name for the attribute in the Attribute Name text box.

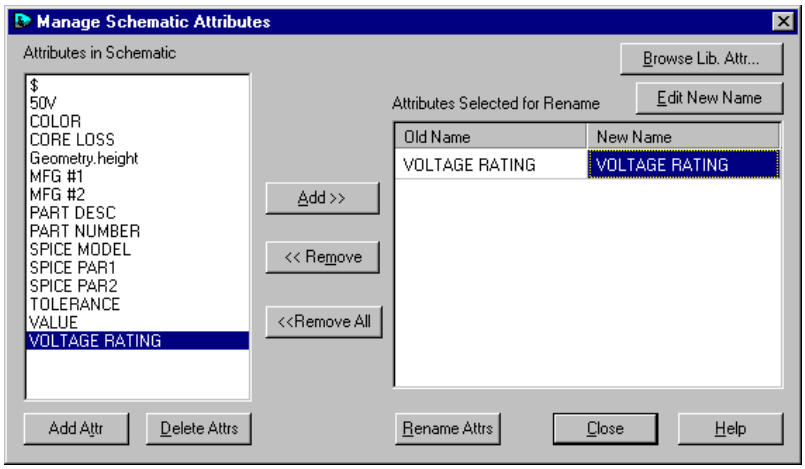
   or

   Choose Browse Lib Attr to choose an attribute already defined in the library. Choose an attribute name from this dialog box and then choose OK.
4. Optionally, type a value for the attribute in the Attribute Value text box.
5. Choose Close to continue.

Manage Schematic Attributes Dialog Box

Use the Manage Schematic Attributes dialog box to manage attributes at the schematic level. You can create a new attribute and automatically assign it to every part in your design. You can also rename an attribute in, or delete an attribute from the schematic. All parts are automatically updated.

Note: This dialog box does not manage attributes in the library. Use the Manage Library Attributes dialog box to manage attributes in your libraries. See the "Managing Attributes in the Library" topic in the "Library Operations" chapter for more information.



Attributes in Schematic

Lists the attributes in the open schematic design file.

Add

Adds selected attributes in the Attributes in Schematic list box to the Attributes Selected for Rename multicolumn list box. Once attributes are listed in the Attributes Selected for Rename multicolumn list box, you can specify a new name for the attribute.

Remove

Removes selected attribute names from the Attributes Selected for Rename multicolumn list box and places them in the Attributes in Schematic list box. Removed attributes are renamed.

Remove All

Removes all attribute names from the Attributes Selected for Rename multicolumn list box and places them in the Attributes in Schematic list box. Removed attributes are not renamed.

Add Attr

Opens the Add New Attribute dialog box where you create a new attribute. The attribute is added to all parts in the open design. See the "Add New Attributes Dialog Box" topic in this chapter for more information.

Delete Attrs

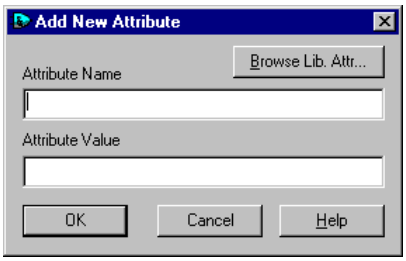
Deletes selected attributes from all parts in the design.

Add New Attributes Dialog Box

Choose Add Attr on the Manage Schematic Attributes dialog box to open this dialog box. Use this dialog box to specify the attribute name and value you want to add to the open design. To create a new attribute:

1. Type a new attribute name in the Attribute Name text box.
2. Type a value for the attribute in the Attribute Value text box.
3. Choose OK.

   If the attribute name already exists in some parts in the design, the message *Items exist with this attribute name - do you wish to overwrite old value with the new name value?* Appears.
4. Choose Yes to add the new name and value to all parts. Choose No to not update the existing attribute. Choose Cancel to return to the Add New Attribute dialog box where you can re-enter a new name and value.



Attribute Name

Type the new attribute name in the text box.

Attribute Value

Type the new attribute value in the text box.

Managing Attributes in a Schematic

Use the Manage Schematic Attributes dialog box to manage attributes at the schematic level. You can rename an attribute in, or delete an attribute from the schematic. All parts are automatically updated.

To rename an existing attribute and apply the change to each part in the design:

1. Choose Tools/Attribute Manager. The Manage Schematic Attributes dialog box appears.
2. Highlight one or more attributes in the Attributes in Schematic list box.
3. Choose Add. The attributes are added to the Attributes Selected for Rename list box.

4. Highlight the attribute in the New Name column and choose Edit New Name or double-click on the attribute.
5. Type a new name for the attribute or choose Browse Lib Attr to choose an attribute already defined in the library. Choose an attribute name from this dialog box and then choose OK.
6. Choose Rename Attrs to rename the attribute and remove it from the Attributes Selected for Rename list box,

To delete an attribute from each part in the design:

1. Choose Tools/Attribute Manager. The Manage Schematic Attributes dialog box appears.
2. Highlight one or more attributes in the Attributes in Schematic list box.
3. Choose Delete.

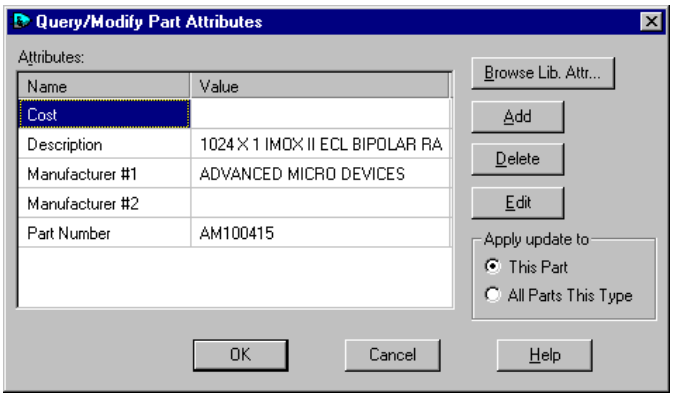
Query/Modify Part Attributes

Use the Attributes button on the Query/Modify Part dialog box to assign or modify part attributes, which is information about the part such as manufacturer and cost. This information travels with the part type information, and you can write out selections to an ASCII file. Attributes consist of two fields: the attribute name, such as Maker, and the attribute value, such as Motorola. See the "Query/Modify Part" topic in the "Design Operations" chapter for more information on the Query/Modify Part dialog box .

1. Choose the Query/Modify icon from the toolbar.
2. Select the part to add or modify the attributes for by selecting a construction line or text string associated with the part.

3. Choose the Attributes icon from the Query/Modify Part dialog box. The Query/Modify Part Attributes dialog box appears.
4. Choose Add. Type a name in the Name column or choose Browse Lib Attr to choose an existing attribute name. Choose an attribute from the list and then choose OK.
5. Type a value for the attribute in the Value column.
6. Choose This Part to only update the selected part or choose All Parts This Type to update all of the parts in the schematic that match the selected part.

Note: Adding or changing attributes in a part at the schematic level does not update the part type. Edit the part in the Part Editor to update the library part.



Browse Lib. Attr

Opens the Browse Library Attributes dialog box that lists all attribute names in all libraries. Choose an attribute to add from this list.

Add

Adds an attribute. Type new attribute names and values in the Name and Value text boxes.

To add an attribute:

1. Choose Add.
2. Type an attribute name in the Name text box.
3. Type an attribute value in the Value text box.

Delete

Removes the selected attribute from the list.

Edit

Modifies an attribute. Select an attribute from the Attributes multicolumn list box, and then choose Edit and type a new name or value in the appropriate text box. You can also double-click on an attribute.

To modify an attribute:

1. Select an attribute from the Attributes list box.
2. Choose Edit. You can also double-click on an attribute.
3. Type a new name or value in the appropriate text box and press Enter.

Apply Update To Area

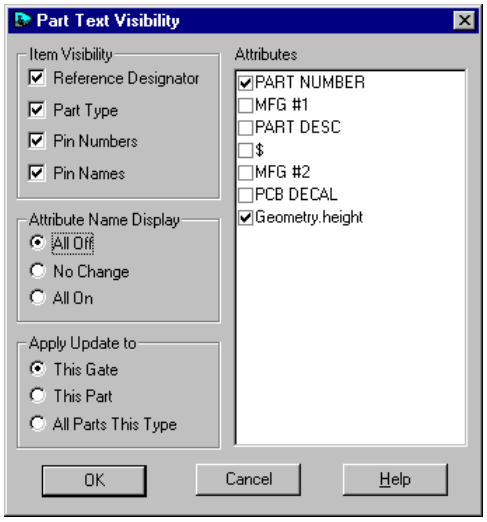
Applies the attribute changes to either the selected part or all parts of the same type.

Query/Modify Part Text Visibility

The Visibility button on the Query/Modify Part dialog box controls the display of text associated with the selected part. You can control the visibility of one part or all parts of the same type. See the "Query/Modify Part" topic in the "Design Operations" chapter for more information on the Query/Modify Part dialog box .

1. Choose the Query/Modify icon from the toolbar.
2. Select a part to modify the text visibility for by selecting the part's outline. The Query/Modify Part dialog box appears.

3. Choose the Visibility icon. The Part Text Visibility dialog box appears.
4. Modify the visibility settings.
5. Choose OK.



Item Visibility

Sets visibility for decal-related text. Turn on the check boxes to display these items in the schematic.

Attributes

Sets visibility for attributes assigned to the selected part. Turn on the check boxes to display these items in the schematic.

Attribute Name Display

Specifies whether to display just the attribute's value or display the attribute name and it's value, for example, PCB DECAL=SO14.

All Off	Makes all attribute names invisible, displays only the value.
No Change	Keeps the current attribute name visibility settings.
All On	Displays all attribute names and their values.

Apply Update To

Controls how parts are updated.

This Gate	Updates the selected gate.
This Part	Updates a part or all associated gates of a part.
All Parts of this Type	Updates all matching gates and/or parts in the design.

Using Attribute Labels

Attribute labels are placeholders for attributes within a CAE Decal. When you add an attribute to a part, it is placed in this reserved location. PowerLogic lets you create an unlimited number of attribute labels. See the "Attributes" topic in this chapter for more information on using attributes in PowerLogic.

Use attribute labels to customize attribute locations in the design. You can create and place attribute labels so that when a part is added to a design, assigned attributes are less likely to exist where a connection or other design information exists. You can also specify a unique justification or orientation for the label.

Note: If you add an attribute and do not have an available attribute label, the attribute is added just below the insertion point of the part. Additional attributes are added below the last one.

Attribute labels do not display in the Schematic Editor or the Part Editor. You must be viewing a decal to view an attribute label. Use Setup Colors to control the display of attribute labels when creating or edit a decal.

You can create an attribute label with a specific name, one that matches the actual attribute's name. This makes placement of specific attribute information easier. You can also use a generic attribute label name, or free label, for any attribute assigned to the part. The name used is *Free Label N, where N is the number of the label.

As you add attributes, they are placed at either the location that matches the attribute name, if specified, or the first available free label location.

Creating Attribute Labels

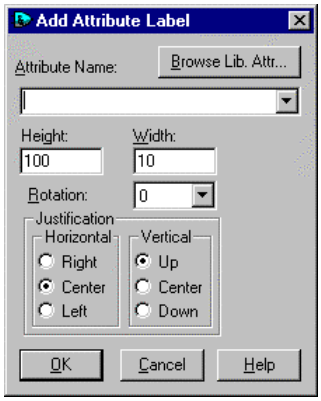
To create attribute labels while editing or creating a CAE Decal:

1. Choose the Drafting toolbox icon from the main toolbar.
2. Choose the Add Attribute Label icon, The Add Attribute Label dialog box appears.
3. Define the attribute label using one of the following methods:
   - Type a name in the Attribute Name drop-down list box for the label that corresponds to an attribute.
   - Choose a name from the Attribute Name drop-down list box. When you choose an attribute name, it is removed from the drop-down list box.
   - Choose *Free Label to use a free label. When you choose Free Label, the number increments to the next available number.
   - Choose the Browse Lib Attr to open the Browse Library Attributes dialog box. Choose an attribute name from the list and then choose OK.
4. Optionally, set the text height, width, or justification.
5. Choose OK.

Note: Use the Display Colors dialog box to turn attribute label display on if necessary.

Add Attribute Label Dialog Box

Use this dialog box to add a new attribute label to a CAE Decal. See the "Creating Attribute Labels" topic in this chapter for more information.



Browse Lib. Attr

Opens the Browse Library Attributes dialog box that lists all attribute names in all libraries. Choose an attribute to add from this list. See the "Browse Library Attributes Dialog Box" topic in the "Library Operations" chapter for more information.

Attribute Name

Type an attribute name or choose a name from the drop-down list.

Note: Attribute names appear only if you are editing an existing decal and have used the Gates icon from the Part Editor to enter the CAE decal. Newly created decals do not display attributes in this drop-down.

Choose and assign a free label. As you assign free labels, the free label number increments for the next time you assign a free label to the same decal.

Height, Width

Changes the text height or line width. Type new values.

Rotation

Changes the orientation of the attribute label in 90-degree increments.

Justification

Choose the insertion point for the attribute label.

Horizontal	Moves the insertion point across the text string for left, right, or center justification.
Vertical	Moves the insertion point up the text string characters for top, center, or bottom of character placement.

Chapter 6
Library Operations

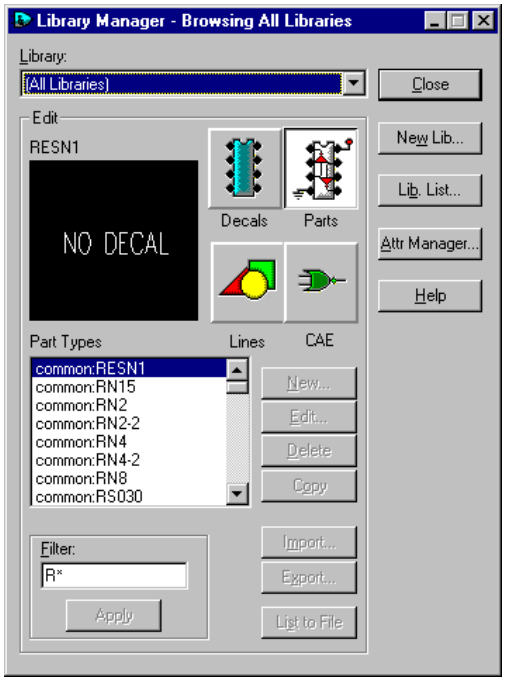
This chapter discusses the library system in PowerLogic and how to use it. Library management is performed in the Library Manager dialog box. This dialog box provides access to part and decal modification as well as logic family assignments and the order of library searches. The topics in this chapter include:

Library Manager
Managing Attributes in the Library

Library Manager

Choose File/Library to open the Library Manager. Use the library Manager to perform operations associated with editing and copying the contents of libraries, such as:

- Creating new libraries.
- Adding, deleting, and modifying library items.
- Printing library contents.
- Importing and exporting library data to ASCII files.
- Choosing which libraries to search.
- Adding attributes to parts.



Selecting a Library

Use the Library list box at the top of the Library Manager dialog box to select a library to edit. You can add to and delete from the library list using the Library List button. See the "Modifying the Library List Order" topic in this chapter for more information.

Library Data Types

A library can contain four kinds of items:

PCB Decal	The graphical representation of the part when it is drawn in the PCB design program. This is often referred to as the *footprint*.
Part Type	Electrical information for the part, such as a 74LS02, including logic family, attributes, pins, and gates.
CAE Decal	The graphical representation of a schematic part, such as a NOR gate.
Lines	General graphical data you can store in the library, such as a company logo, to use in any .sch file.

Libraries store decal and part type attributes and attribute labels, but they do not store attribute values. See the "Working with Attributes" chapter for more information on using attributes in PowerLogic.

General Purpose Library Operations

Use the following command buttons to perform general-purpose library operations:

New Lib	Initializes a library. You can create a new library or reinitialize an existing library. You cannot reinitialize a library that is marked as read-only.
Lib. List	Opens the Library List dialog box, which lets you specify libraries to search for decals, parts, and line items.
Attr Manager	Opens the Manage Library Attributes dialog box that lists the attribute names in your libraries and lets you add an attribute to, rename an attribute in, or delete an attribute from a library. See the "Creating Attributes in the Library" topic in this chapter for more information. When you add an attribute in this dialog box, each part in the current library directory is updated to contain the new attribute
Close	Exits the Library Manager.

Edit Operations

Use the controls in the Edit section to view and modify objects in the library.

1. Choose the appropriate button to select which types of objects to access: decals, part types, lines, or CAE decals. The Part Types list box contains the names of all objects of that type in the current library.
2. Use the Filter to limit the items that appear in the list box. See the "Search Filter" topic in this chapter for more information.
3. Use one of the command buttons described below for the selected objects.

Note: When you select an object, its picture appears above the Part Types list box. Parts are drawn using the first assigned decal.

New

Creates a new object. You must provide a name for the object.

- For Decals, the currently assigned PCB decal appears. The New and Edit buttons gray. You can only copy or delete the decal. Use the Library Manager in the PCB design program to modify the PCB decal.
- For Parts, the Part Information dialog box appears. See the "Part Information for Part" topic in the "Part Editor Operations" chapter for more information.
- For Lines, any 2D drafting items appear, for example, drawing formats, title blocks, etc. The New and Edit buttons gray. You can only copy or delete the drafting items.
- To add new drafting items to the library, create and combine the items in PowerLogic, then use the Save to Library icon from the Drafting toolbox. See the "Drafting Operations" chapter for more information.
- For CAE decals, the Part Editor opens.

Edit

Modifies an existing object.

- For Decals, the currently assigned PCB decal appears. The New and Edit buttons gray. You can only copy or delete the decal. Use the Library Manager in the PCB design program to modify the PCB decal.
- For Parts, the Part Information dialog box appears. See the "Part Information for Part" topic in the "Part Editor Operations" chapter for more information.
- For Lines, any 2D drafting items appear, for example, drawing formats, title blocks, etc. The New and Edit buttons gray. You can only copy or delete the drafting items.
- To edit drafting items in the library, use the Add from Library icon from the Drafting toolbox to add the object to PowerLogic. Uncombine the object, modify the object, use Combine to combine the items, and then use the Save to Library icon from the Drafting toolbox. See the "Drafting Operations" chapter for more information.
- For CAE decals, the Part Editor opens.

Delete

Deletes selected objects. You can select and delete multiple objects at the same time.

Copy

Copies an existing object, giving it a new name. You cannot copy CAE decals.

Import

Imports library data from a previously exported ASCII file. You must supply the filename.

Export

Exports one or more selected items to an ASCII file. You can then import this file to another library. A prompt asks you to supply the filename.

List to File

Produces a report about the selected items. If you select a read-only library, the New, Edit, Delete, Copy, and Import buttons gray.

Search Filter

Several dialog boxes contain a Filter area. Use Filter to search the selected library for a specific item name or for names that match a wildcard text string.

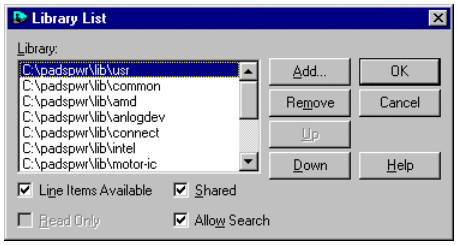
Use the Library drop-down list to choose library directories. See the "Preface" for more information.

Modifying the Library List Order

When you add a part to the design, you are actually retrieving information about that part from the library. Use the Library List dialog box to tell the program which libraries to search and in what order.

Choose File/Library and choose Lib. List on the Library Manager dialog box.

The Library List dialog box contains a list of libraries to search. The libraries are searched in the order in which they are listed, beginning with the first library in the list.



Library

Lists the libraries to search. The libraries are searched in the order in which they are listed, beginning with the first library in the list.

Add

Adds a new library to the search list. When you choose Add, the Add Library dialog box appears. Choose a library to open. The library is added to the Library list immediately below the currently selected library. If a library is not selected, the new library is added to the bottom of the search list.

Remove

Deletes the selected library from the Library list.

Up

Moves the selected library up one position in the search order. The libraries are searched in the order in which they are listed, beginning with the first library in the list.

Down

Moves the selected library down one position in the search order. The libraries are searched in the order in which they are listed, beginning with the first library in the list.

Line Items Available

Allows use of line items. Turn on this library option if the library contains Lines graphical data.

Shared

Shares the library across a networked environment, allowing more than one user to simultaneously access the library. Turn on this library option if you want to share the library.

Read Only

Sets the library as read-only. Turn on this library option if you do not have permission to write to the library file.

Note: The Read Only check box is always checked and grayed if the library files are set as Read Only in the Microsoft Windows File Manager. Read Only is an attribute that is set on files using the Windows File Manager. The system administrator who owns the files is the only one who can control this process; this ensures file protection.

Allow Search

Searches the library when performing operations that involve libraries, such as adding parts. Turn on this library option if you want PowerPCB to search the library.

Managing Attributes in the Library

Use the Manage Library Attributes dialog box to create new attributes, to list the attribute names in your libraries and to add an attribute to, rename an attribute in, or delete an attribute from the library. See the "Managing Attributes in a Schematic" topic in the "Working with Attributes" chapter for more information on managing attributes in an open design file.

Creating Attributes in the Library

You can create an attribute that is assigned to every part in a library. See the "Working with Attributes" chapter for more information on creating attributes in PowerLogic.

To create a new attribute in a library:

1. Choose File/Library. The Library Manager dialog box appears.
2. Choose Attr Manager. The Manage Library Attributes dialog box appears.
3. Choose a library from the Library drop-down list box, or choose (All Libraries) to add the attribute to all libraries.
4. Choose Add Attr. The Add New Attribute to Library dialog box appears.
5. Type a name for the attribute in the Attribute Name text box.

   or

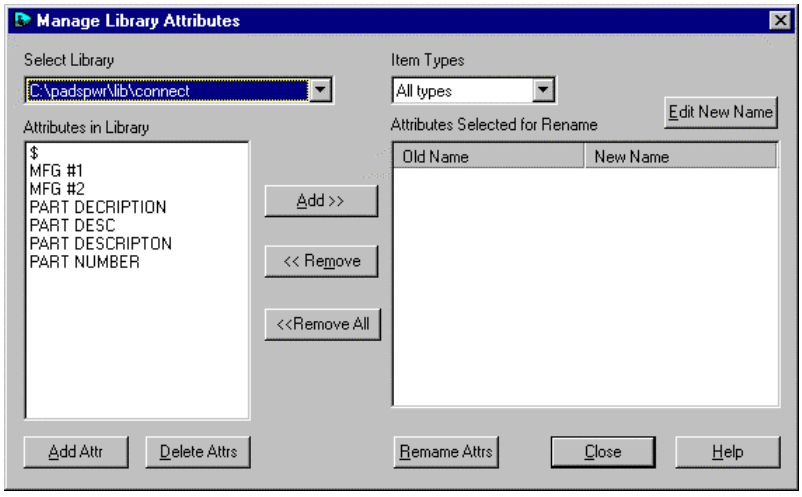
   Choose Browse Lib Attr to choose an attribute already defined in a different library. Choose an attribute name from this dialog box and then choose OK.
6. Optionally, type a value for the attribute in the Attribute Value text box.
7. Choose Close.

Manage Library Attributes Dialog Box

Choose Attr Manager on the Library Manager dialog box to open this dialog box. Use this dialog box to list the attribute names in your libraries and to add an attribute to, rename an attribute in, or delete an attribute from the library.

When you add an attribute in this dialog box, each part in the current library directory is updated to contain the new attribute. See the "Creating Attributes in the Library" topic in this chapter for more information.

Note: This dialog box does not manage attributes in the current design. Use the Manage Schematic Attributes Dialog Box to manage attributes in an open design file. See the "Creating New Attributes in a Schematic" topic in the "Working with Attributes" chapter for more information.



Select Library

Specifies the library with which to work. You can select an individual library or all libraries.

Item Types

Specifies the attributes to list in the Attributes in Library list box.

Edit New Name

Allows you to edit the selected New Name cell. You can also double-click on the New Name cell to edit it. This button grays until you rename an attribute. You must select a New Name cell to activate this button.

Attributes in Library

Lists the attributes in the selected library.

Attributes Selected for Rename

Lists the attribute you want to rename and its new name.

Old Name	Specifies the existing attribute name.
New Name	Specifies the new name of the attribute.

Add

Adds selected attributes in the Attributes in Library list box to the Attributes Selected for Rename multicolumn list box. Once attributes are in the Attributes Selected for Rename multicolumn list box, you can specify a new name for the attribute.

Remove

Removes selected attribute names from the Attributes Selected for Rename multicolumn list box and places them in the Attributes in Library list box. Removed attributes are not renamed.

Remove All

Removes all attribute names from the Attributes Selected for Rename multicolumn list box and places them in the Attributes in Library list box. Removed attributes are not renamed.

Add Attr

Opens the Add New Attribute to Library dialog box where you create a new attribute.

Delete Attrs

Deletes selected attributes from the library.

Rename Attrs

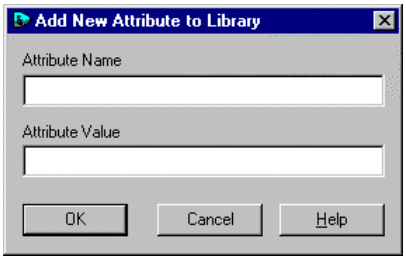
Starts the rename process for attributes in the Attributes Selected for Rename multicolumn list box. Renamed attributes are removed from this list box and added to the Attributes in Library list box.

Add New Attribute to Library Dialog Box

Choose Add Attr on the Manage Library Attributes dialog box to open this dialog box. Use this dialog box to specify the attribute name and value you want to add to the selected library.

See the "Creating Attributes in the Library" topic in this chapter for more information on attributes in the library.

Note: If the attribute name already exists in some parts in the design, the message *Items exist with this attribute name - do you wish to overwrite old value with the new name value?* appears. Choose Yes to add the new name and value to all parts. Choose No to not update the existing attribute. Choose Cancel to return to the Add New Attribute to Library dialog box where you can re-enter a new name and value.



Attribute Name

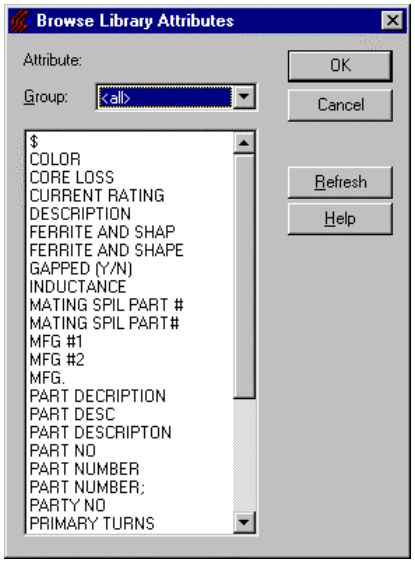
Type the new attribute name in the text box.

Attribute Value

Type the new attribute value in the text box.

Browse Library Attributes Dialog Box

Use this dialog box to browse and list all attribute names from libraries specified in the Library List dialog box.



Attribute Name

Lists the selected attribute name.

Group

Allows you to choose an attribute group to view. If you assign multiple attributes and some of those are structured attributes, this drop-down list box acts as a filter and allows you to choose the attribute group to view.

Library Attribute List

Lists the attributes for the libraries in your library list. This includes both part type and decal attributes. Attribute values are not included in the list.

Refresh

Resets the contents of the library attribute list box based on the Library List dialog box.

Chapter 7
Part Editor Operations

This chapter discusses operations in the Part Editor. You access the part editor from the Library Manager. See the "Library Operations" chapter for more information. This chapter covers the following topics:

The Part Editor
Creating Single Gate Parts
Creating Multigate Parts
Part Information for Part
Creating New Parts from Existing Parts
Decal Wizard
Terminal Toolbox
Special Symbols

The Part Editor

Use the Part Editor to create and modify standard schematic symbols such as gates, resistors, capacitors, connectors, etc., and special symbols such as ground, power and off-page references. You also use the Part Editor to create new CAE decals, pin decals, and assign PCB decals; the physical representation of the part in a PCB design system.

Choose Tools/Part Editor to enter PowerLogic's Part Editor. Unneeded menu commands and toolbar icons are removed or replaced with menu commands and toolbar icons specific to creating and modifying parts.

A PowerLogic part is comprised of three elements:

Part Type	The electrical information for a part. This includes the logic family, CAE Decal assignment, part name, connection pins, gate swapping information, etc.
CAE Decal	The CAE decal is the logic symbol that appears on the schematic; for example, a NAND gate.
PCB Decal	The graphical representation of the part when it is drawn in the PCB design program. It is often referred to as the footprint. Assigning a PCB Decal is optional and only required if you are passing a netlist to PowerPCB or Perform.

Changing and Updating Library Parts

When you add a part to a schematic, a copy of the library part is made and added to the schematic database. Thereafter, the schematic representation is uncoupled from the library. Modifying a part in the Part Editor only effects the library entry of that part, not the current version in the schematic.

See the "Query/Modify Part" topic in the "Design Operations" chapter for information on updating parts on the schematic with the modified version.

See the "Updating Off-page from Library" and "Updating Pins from Library" topics in this chapter for information on updating special symbols on the schematic with those from the library.

Creating Single Gate Parts

Below is the basic procedure for creating a single gate part in PowerLogic. Single gate parts are parts that are represented by a single schematic symbol; for example, an application-specific integrated circuit.

There are many steps used to create a new part, but there are dialog boxes and information windows to help you. Refer to the referenced documentation in the steps below for additional information.

In the following steps, you create the CAE decal that shares the same name as the part type before entering electrical information. If you want to use an existing CAE decal instead of creating a new one, enter the electrical information first. See the "Creating Multigate Parts" topic in this chapter for the sequential steps.

1. Choose Tools/Part Editor to enter the Part Editor. PowerLogic enters part creation mode.

2. Choose Edit/Gate, or choose the Edit Gate Decal icon from the toolbar.
3. Choose OK to create a decal for the new part.

   Four text entries appear in the work area:

U999	The default space reserved for the reference designator in the schematic.
Part Name	The default location of the part type name in the schematic.
* Free Label 1	The default location for the first attribute label. See the "Using Attribute Labels" topic in the "Working With Attributes" chapter for more information on creating additional attribute labels.
* Free Label 2	The default location for the second attribute label.

4. Construct the new CAE decal for the part manually or use the Decal Wizard automatic part creation tool. See the "Constructing the Decal" topic in this chapter for more information.

5. After you create the decal for the part, assign electrical information to it. See the "Assigning Electrical Information to the Part" topic in this chapter for more information.
6. Save the part to the library. See the "Saving the Part to the Library" topic in this chapter for more information.

Constructing the Decal

To manually construct the decal:

1. Choose the Drafting Toolbox icon from the toolbar. See the "Drafting Operations" chapter for more information.

2. Choose the Create 2D Line icon from the Drafting toolbox, and then construct the body of decal.

3. Choose the Terminal Toolbox icon from the toolbar. See the "Terminal Toolbox" topic in this chapter for more information.

4. Choose the Add Terminal icon from the Terminal toolbox. See the "Adding Terminals" topic in this chapter for more information.
5. Indicate the locations for the terminals.
6. Choose Cancel from the pop-up menu when you finish adding terminals.

7. Choose the Set Pin Number icon from the Terminal toolbox. See the "Setting Pin Numbers" topic in this chapter for more information.
8. Type the starting number in the information window.
9. Choose OK.

10. Select each pin or pin text in ascending sequence to assign a number. You can click twice on a pin to skip a number. Use the Change Pin Number icon from the Terminal toolbox to change a pin number if necessary. See the "Changing Pin Numbers" topic in this chapter for more information.
11. Use the other options of the Terminal toolbox to modify the pin numbers and add pin names, pin swapping information, pin type, etc.
12. Choose File/Return to Part. Answer Yes to the *Keep changes to gate?* prompt.

To automatically construct the decal:

1. Choose the Decal Wizard icon from the Drafting toolbox to open the Decal Wizard. See the "Decal Wizard" topic in this chapter for more information.
2. Use the Decal Wizard to create the outline and add terminals.
3. Use the Terminal toolbox to modify the pin numbers created by the Decal Wizard and add pin names, gate swapping information, etc. See the "Terminal Toolbox" topic in this chapter for more information.
4. Choose File/Complete.

Assigning Electrical Information to the Part

1. Choose Edit/Electrical, or choose the Edit Electrical icon from the toolbar. The Part Information dialog box appears. See the "Part Information for Part" topic in this chapter for more information.
2. Use Pin Count to indicate the total number of pins in the part. This is mandatory if there are signal pins in the part that were not added when you created the CAE decal in the steps above.

3. Optionally, choose the Decals icon from the Part Information dialog box to assign the PCB decal. The pin count for the assigned PCB decal must be equal to or greater than the electrical description.
4. Choose a Logic Family. See the "Part Information for Part" topic in this chapter for more information.
5. Optionally, choose Alpha Pins if the part uses alphanumeric characters.
6. Optionally, choose Signal Pins to assign power and ground signal information.
7. Optionally, choose Attributes to assign bill of material and part list information.
8. Choose OK. The Part Editor now displays the CAE Decal.

Saving the Part to the Library

Use the Save Part and Gate Decals As dialog box to save part types to the library. You can also rename the associated decal during the save process. If you change the pin count or other information in the decal, this prevents other parts that use the same decal from being effected.

Note: If you decrease the number of pins in the associated decal, the Change Type option in the Query/Modify Part dialog box does not update the schematic copy of the part with the modified version in the library .

To save a part type:

1. Choose File/Save.
2. Type a name in the Name of Part text box.
3. Choose a library location from the Library drop-down list box.
4. To change the name of the associated decal:
   a. Select the decal from the CAE Decal column of the Names of Decals multicolumn list box and choose Edit.
   b. Type a new name for the decal.
5. Choose OK.

If the part type already exists, the message *Part Type item exists. Overwrite item Y/N?* appears. Choose Yes to overwrite the part, or No to cancel the save.

If you modified the part using Edit Part/Hierarchical Symbol to enter the Part Editor, the message *Update all parts of Part Type* appears. Choose Yes to update parts on the schematic with the revised library version, or No to leave the schematic version of the part unchanged.

Library Management for Saved Part Types

The library directory used for part types and associated decals is described below.

- Part type data is saved under the name and library chosen in the Save Part and Gate Decals As dialog box.
- Decals created when editing the part type are saved in the same library as the part type.
- Decals that are modified but not renamed are saved to the library from which they were opened.
- Renamed decals, whether modified or not, if unique across all libraries, are saved in the same library as the part type.
- Renamed decals that are not unique across all libraries display the overwrite warning prompt. If you choose Yes to overwrite an existing decal with the renamed decal, it replaces the library copy in its current library. If you answer No to the prompt, you can choose a different decal name and directory.

Saving a Modified Part Type with a Different Name

If you modify a part type and want to keep the existing library copy, use the Save Part and Gate Decals As dialog box to rename the part. After modifying the electrical information:

1. Choose File/Save As. The Save Part and Gate Decals As dialog box appears.
2. Type a new name for the part type in the Name of Part text box.

   Optionally, use the Library drop-down list box to specify a new library location. See "Library Management for Saved Parts" in this chapter for more information.

Saving a Modified Decal with a Different Name

If you modify a decal associated with a part type, use the Save Part and Gate Decals As dialog box to rename the decal. After modifying the decal:

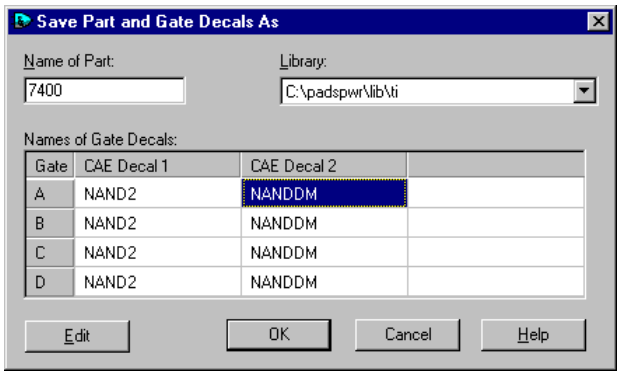
1. Choose File/Return to Part. Answer Yes to the *Returning to Part level - Keep changes to Gate* prompt.
2. Choose File/Save As. The Save Part and Gate Decals dialog box appears.
3. Select the decal in the Name of Gate Decal multicolumn list box and choose Edit.

4. Type a new name for the Decal and choose OK.
5. Answer Yes to the *Part Type item exists. Overwrite item?* prompt to replace the current library part with the one containing the modified decal.

Note: See the "Saving a Modified Part Type with a Different Name" topic in this chapter for information on saving a part type with a different name.

Save Part and Gate Decals As Dialog Box

The Save Part and Gate Decals As dialog box saves a new or modified part type to the library. You can also rename a modified decal to prevent other parts that use the same decal from being effected.



Name of Part

Defines the part type name in the library.

Library

The library to which directory information is saved.

Name of Gate Decals

Displays the name of decal and any alternate decals associated with the part type. Select a decal name and choose Edit to rename the decal.

Edit

Makes the highlighted field in the multicolumn list box editable. You can also double click a text field to edit it.

Creating Multigate Parts

Listed below is the basic procedure for creating a multigate part in PowerLogic. Multigate parts are parts that use more than one symbol or gate to represent a complete part in the PCB design system; for example, a 7400.

There are many steps used to create a new part, but there are dialog boxes and information windows to help you. Refer to the referenced documentation in the steps below for additional information.

1. Choose Tools/Part Editor to enter the Part Editor.

2. Choose Edit/Electrical, or choose the Edit Electrical icon from the toolbar. The Part Information dialog box appears. See the "Part Information for Part" topic in this chapter for more information.
3. Use Pin Count to indicate the number of pins in the part. Enter the number of pins for the entire package including signal pins, not for an individual gate.

4. Optionally, choose the Decals icon from the Part Information dialog box to assign the PCB decal. The pin count for the assigned PCB decal must be equal to or greater than the electrical description.

5. Choose the Gates icon from the Part Information dialog box. The Gate Information for Part dialog box appears.
6. Choose Add Gate, once for each gate in the part. The gates are listed alphanumerically in the Gates and Pins area. See the "Gates Tab" topic in this chapter for more information.
7. Highlight a gate, and then choose Assign Decals. The Assign Decal to Gate dialog box appears. See the "PCB Decals Tab" topic in this chapter for more information.
8. Set the filter and select a symbol from the Unassigned Decals area.
9. Choose Assign. Choose additional CAE decals if you want to assign alternate decals.
10. Optionally, assign gate swapping information for the part from the Gate Information for Part dialog box.
11. Repeat steps 7 through 10 for the other gates.
12. Choose OK.
13. Choose a Logic Family, Prefix List, and other electrical information.
14. Choose OK. The Part Editor displays the assigned CAE Decal outlines. Note that no pin information appears.
15. Assign pin information to the CAE decal. See the "Assigning Pin Information to the CAE Decal" topic in this chapter for more information.
16. Save the part to the library. See the "Saving the Part to the Library" topic in this chapter for more information.

Assigning Pin Information to the CAE Decal

1. Choose Edit/Gate, or choose the Edit Gate Decal icon from the toolbar. The Select Gate Decal dialog box appears. See the "Select Gate Decal" topic in this chapter for more information.
2. Choose a gate from the Gate drop-down list box, or select a gate from the work area. The decal appears in the CAE Decal Editor.

3. Choose the Terminal Toolbox icon from the toolbar. See the "Terminal Toolbox" topic in this chapter for more information.

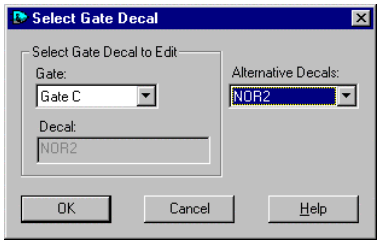
4. Choose the Set Pin number icon from the Terminal toolbox. See the "Setting Pin Numbers" topic in this chapter for more information.
5. Type the starting number in the information window.
6. Choose OK.

7. Select each pin or pin text in ascending sequence to assign a number. You can click twice on a pin to skip a number. Use the Change Pin Number icon from the Terminal toolbox to change a pin number if necessary. See the "Changing Pin Numbers" topic in this chapter for more information. Any alternate CAE decals are assigned the same pin numbers.
8. Set other pin information such as pin type and pin swapping, as needed.
9. Choose Complete.
10. Repeat steps 1 through 9 for the other gates.

Select Gate Decal

Use the Select Gate Decal dialog box to choose a gate to assign pin information to pin number, pin type, swapping information, etc. To open this dialog box, you must first assign gates to a part through the Gates option of the Part Information dialog box. See the "Creating Multigate Parts" topic in this chapter for the proper sequence.

You can also create new alternate CAE decals and assign them to the gate. See the "Gates Tab" topic in this chapter for information on assigning existing CAE decals as alternates.



Gate

Selects a gate to edit. You can also select the gate from the work area.

Decal

Lists the currently assigned CAE decal. You can also type the name of a new alternate decal here.

Alternative Decal

Edits an alternate decal assigned to a gate or creates a new alternate decal. You can also select the alternate from the work area.

To create a new alternate decal:

1. Choose <New Decal> from the drop-down list box.
2. Type a name for the new decal in the Decal text box.
3. Choose OK from the information window.
4. Create the new decal.

Creating a New CAE Decal

Creating a CAE Decal follows the same basic steps as creating a new packaged part. The following steps place PowerLogic directly into decal editing mode:

1. Choose Tools/Part Editor to enter the Part Editor.
2. Choose File/New. The Select Type of Editing Item dialog box appears. See the "Select Type of Editing Item" topic in this chapter for more information.
3. Choose CAE Decal.
4. Choose OK.

   Four text entries appear in the work area:

U999	The default space reserved for the reference designator in the schematic.
Part Name	The default location of the part type name in the schematic.
* Free Label 1	The default location for the first attribute label. See the "Using Attribute Labels" topic in the "Working with Attributes" chapter for more information on creating additional attribute labels.
* Free Label 2	The default location for the second attribute label.

5. Construct the new CAE decal for the part manually or use the Decal Wizard automatic part creation tool. See the "Constructing the New Part" topic in this chapter for more information.
6. Save the decal to the library. See the "Saving the Part to the Library" topic in this chapter for more information.

Constructing the New Part

To manually construct the new part:

1. Choose the Drafting Toolbox icon from the toolbar. See the "Drafting Operations" chapter for more information.

2. Choose the Create 2D Line icon from the Drafting toolbox, and then construct the body of decal. See the "Drafting Operations" chapter for more information.

3. Choose the Terminal Toolbox icon from the toolbar. See the "Terminal Toolbox" topic in this chapter for more information.

4. Choose the Add Terminal icon from the Terminal toolbox. See the "Adding Terminals" topic in this chapter for more information.
5. Indicate the locations for the terminals.
6. Choose Cancel from the pop-up menu when you finish adding terminals.
7. Optionally, use the other options of the Terminal toolbox to add additional terminals, change the pin decal, or change the pin sequence number. The other options in the Terminal toolbox gray when you create a CAE Decal.

To automatically construct the decal:

1. Choose the Decal Wizard icon from the Drafting toolbox to open the Decal Wizard. See the "Decal Wizard" topic in this chapter for more information.
2. Use the Decal Wizard to create the outline and add terminals.
3. Use the Terminal toolbox to modify the pin numbers created by the Decal Wizard and add pin names, gate swapping information, etc. See the "Terminal Toolbox" topic in this chapter for more information.
4. Optionally, use the Terminal toolbox to add additional terminals, change the pin decal, or change the pin sequence number. The other options in the Terminal toolbox gray when you create a CAE Decal.
5. Choose File/Complete.

Part Information for Part

Use the Part Information dialog box to add or modify electrical information for a part. Electrical information includes the number of pins in the part, the logic family, the PCB decal assignment, the CAE decal assignment, signal pins, gate swapping information, etc. You must identify the electrical or part type information, before assigning a CAE Decal to the part.

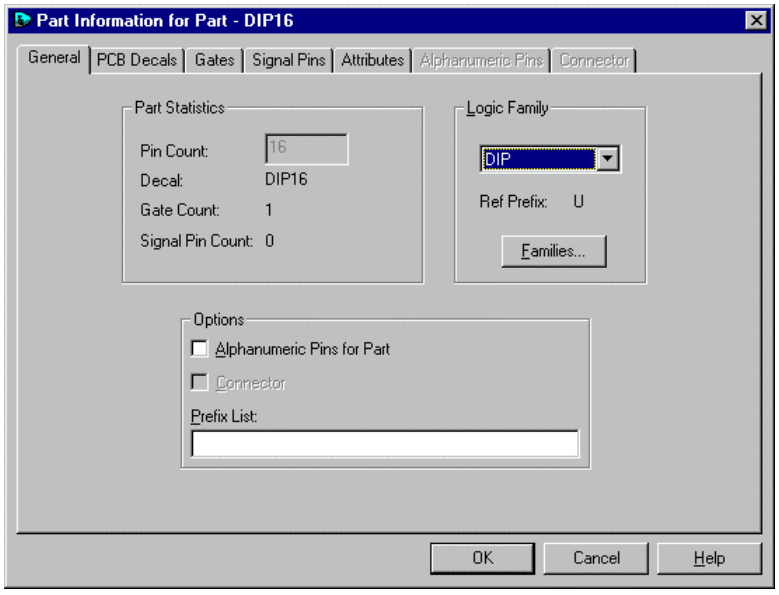
To use the Part Information dialog box:

1. Choose the Edit Electrical icon from the toolbar in the Part Editor when creating or modifying a Part Type. The Part Information dialog box appears.
2. Choose a tab on the dialog box to set information related to the tab name. The available tabs are: General, PCB Decals, Gates, Signal Pins, Attributes, Alphanumeric Pins, and Connector.

3. When you choose the tab, the options for that tab appear. Set the options you want to change.
4. Choose other tabs to set other options as necessary.
5. When you finish setting options, choose OK. The changes are saved and the Part Information dialog box closes. Choose Cancel to close the Part Information dialog box without saving any of the changes.

General Tab

Use the General tab in the Part Information dialog box to view part statistics and set family information.



Part Statistics

Lists information about the part you are creating or editing under the following categories:

Pin Count	Sets the number of pins in the part. You must indicate the number of pins before assigning other information.
Decal	Lists the name of the decal.
Gate Count	Lists how many gates exist in the part.
Signal Pin Count	Lists the number of signal pins in the part.

Logic Family

Identifies the Logic Family. Choose a Logic Family from the list box. When you choose a Logic Family, the default reference designator prefix for this part appears next to Ref. Prefix.

Choose Families to open the Logic Families dialog box where you can add, delete, and edit logic families and default reference designator prefixes.

Alphanumeric Pins for Part

Allows the assignment of alphanumeric names to pins (A, B, C, etc.) or an alphanumeric prefix (A1, A2, A3, etc.). You can select the Alphanumeric Pins tab to assign names. See the "Alphanumeric Pins Tab" topic in this chapter for additional information.

Connector

This check box is automatically on when you create or modify connectors. This check box grays if you open a part other than a connector. See the "Connector Tab" topic in this chapter for more information.

Prefix List

Sets the prefix for the part using the Prefix List text box. Searching by part prefix allows you to edit a part, for example 7400, and apply those edits to all variations of 7400 searched for in the Prefix List.

To use a prefix, type the prefix using a question mark (?) as a wildcard character; to indicate any character for which you want to search. You can search using multiple variations in the prefix; for example, you can set the 7402 family prefix specification to the following:

?4 ?4HC ?4HCT ?4AS ?4ALS ?4AS ?4LS

Any edits you make apply to all of the parts contained within the above prefix string.

A shorthand description for 5400 and 7400 parts can be created using a prefix and a suffix. The suffix defines an entire family of ICs to be stored under one part type. This enables you to include all variations of the family with a single entry in the library. The first part of the suffix is a backslash (\) followed by the number series that uniquely identifies the device family; for example, \00 for a 7400, 74LS00, 54L00; a \192 for 74192, 74HCT192, and so on.

Editing Logic Families

Use the Logic Families dialog box to add, delete, and modify logic family names and default reference designator prefixes.

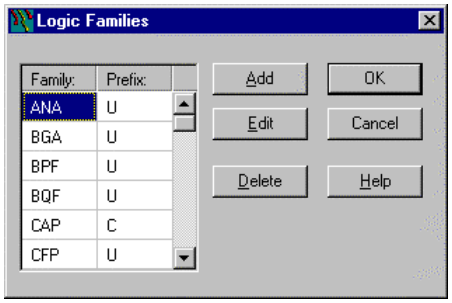
To open the Logic Families dialog box, choose Families from the Part Information dialog box. The Logic Family dialog box lists all existing logic families and default reference designator prefixes.

To change the name or prefix of an existing Logic Family:

1. Select the logic family from the Families list box.
2. Type a new name in the Family text box.
3. Type a new prefix to associate with the Logic Family in the Prefix text box.

To add a new Logic Family:

1. Choose Add. The Add Logic Family dialog box appears.



2. Type the new Logic Family name and choose OK.
3. Type the prefix you want to use for the new family in the Prefix text box.

Use the Logic Families dialog box to add, delete, or modify logic family names and default reference designator prefixes.

Choose the Families button in the Parts Information dialog box to display this dialog box. See the "Part Information for Part" topic in this chapter for additional information.

Family and Prefix

Lists all existing logic families and their associated prefixes. You can type a new family name or a new prefix for the family. To modify the family or prefix, select the item from the multicolumn list box and choose Edit. You can also double-click an item to edit it.

Add

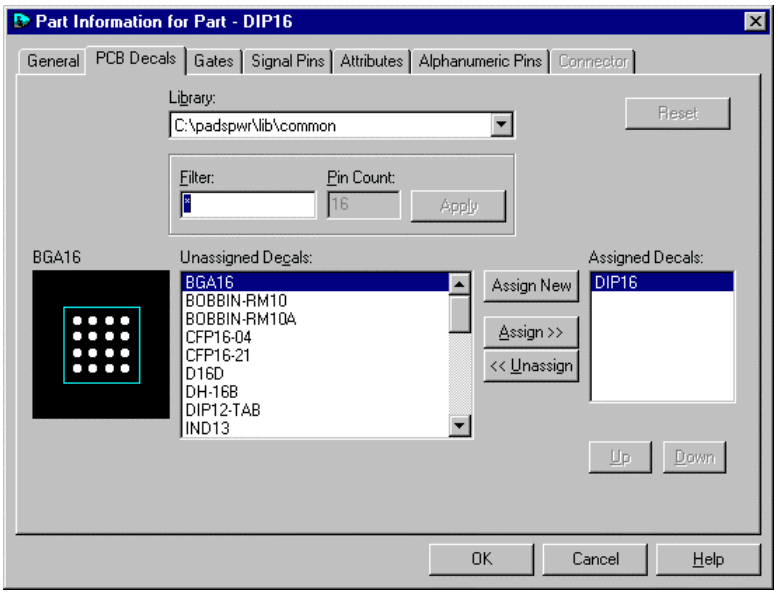
Adds a new logic family to the part. Provide a name and prefix for the new family.

Delete

Deletes the family selected in the multicolumn list box.

PCB Decals Tab

Use the PCB Decals tab on the Part Information dialog box to assign decals, or *footprints*, to library parts. This information is used to load the proper parts when you import a netlist into a PCB layout application. You can assign up to 16 PCB decals to a part; one as the default decal and the others as alternates.



Assigning a PCB Decal is one way of determining the number of pins in a part. The Pin Count text box on the PCB Decals tab grays when you assign a PCB decal, unless you create a connector. See the "Part Information for Part" topic in this chapter for additional information on the Pin Count option.

To assign decals:

1. Select one or more decal names.
2. Choose Assign.

Note: As a shortcut, double-click on a decal name to assign it to the part.

You can assign up to 16 decals. All assigned decals must have the same number of pins.

Library

Lists all libraries. Choose the library from which you want to assign decals. The available decals from the selected library appear in the Unassigned Decals list box. The list changes according to the library you select.

Filter

Limits the items that appear in the Unassigned Decals list box. Type the prefix you want to filter decals by, followed by an asterisk. For example, type **DIP*** to view all DIP decals. Type * to view all decals.

Choose Apply to see the results of your search in the Unassigned Decal list box.

Pin Count

Contains the number of pins in the decal.

Preview Area

Displays the decal that you select in the Unassigned Decals list box.

Unassigned Decals

Lists all decals from the selected library that you can assign to the part.

Assigned Decals

Lists all decals assigned to the part. You can assign up to four decals. Assigned decals must have the same number of pins. The first decal is the default decal. Use the Up and Down buttons to reorder the assigned decals.

Assign New

Assigns the decal name of a nonexistent decal in the library. This feature is very beneficial for schematic designers. You can specify a decal that doesn't exist in your library, but may exist in the PCB designers library.

When you choose Assign New, the Assign New PCB Decal dialog box appears. Type a name and choose OK. The name is added to the end of the Assigned Decals list. To unassign the new decal name, choose Unassign.

Assign

Assigns decals. To assign decals to the part:

1. Select one or more decal names from the Unassigned Decals list box.
2. Choose Assign. The decal moves to the Assigned Decals list box; the decal is assigned to the part.

Note: As a shortcut, double-click on a decal name to assign it to the part.

You can assign up to four decals. All assigned decals must have the same number of pins.

Unassign

Removes decals from the Assigned Decals list box and returns them to the Unassigned Decals list box. To delete a decal assignment:

1. Select a decal name from the Assigned Decals list box.
2. Choose Unassign. The decal moves to the Unassigned Decals list box; the decal is not assigned to the part.

Up and Down

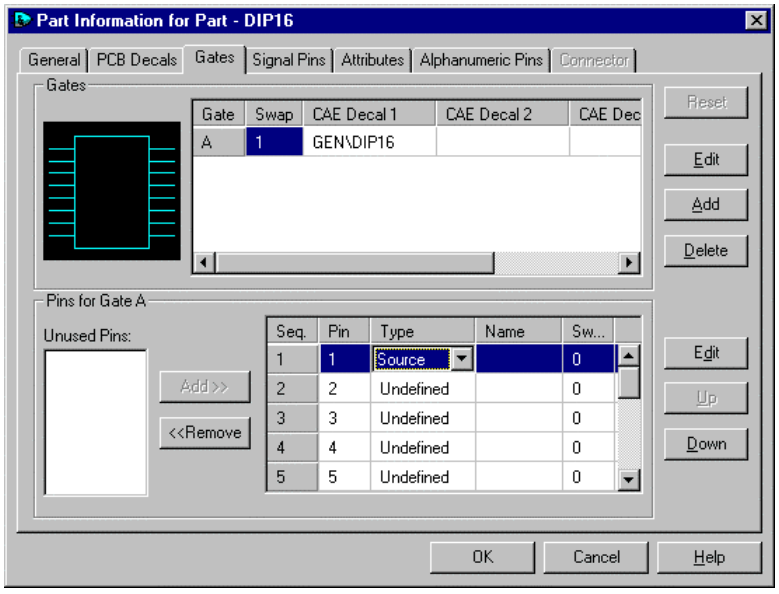
Changes the order of assigned decals in the Assigned Decals list box. The first decal is the default decal. Use Up and Down to move the decal you want to use as the default decal to the top of the Assigned Decals list box.

Reset

Returns the options on the current tab to their original settings; the settings that appeared when you first opened the tab. Reset only resets the current tab.

Gates Tab

Use the Gates tab to assign gate information to a part, including the number of gates, pin assignments for each gate, and swap capability information for both. You also use this dialog box to assign a CAE decal to a part.



Gate Decal and Alternates

For each gate, you must enter the CAE decal name; the name of the logic symbol that is used to represent the part in the schematic. Alternate decal assignments must have the same number of pins. One primary and three alternate decals may be defined for each gate.

Gate and Pin Swap Information

If at least one decal is assigned to a part, you can enter or modify its gate information. This includes swap enable or disable for gates within a part or between similar parts. This information is used in the PCB design system when swapping is performed after placement for connection length minimization and routing optimization.

Pins with like numbers can swap within a gate. Gates with like numbers can swap within the part, or to other similar parts. A one "1" in the Swap column indicates that the gate is swappable with gates of the same part type in the PCB design. If a part contains more than one type of swappable gate, identify the second type with the number 2, the third type with 3, etc. The number 0 indicates that no swapping can occur.

You can assign pin type information in the Pins for Gate area; for example, Source or Load. You can also assign a name to a pin for documentation purposes. The pin name is not required.

Gates Multicolumn List Box

Lists the gates defined for the part type.

Gate	Lists the gate for which you define swap IDs and alternate decals. You cannot modify the gate name.
Swap ID	Swaps gates with the same Swap ID number. If it is valid to swap the selected gate with another gate in the list, assign a Swap ID to the gate. To disable swap for the gate, type a value of 0. Double-click the Swap entry to edit or select the Swap entry and choose Edit. Type a Swap ID between 0 and 100 in the text box.
CAE Decal 1-4	Define up to four alternate Logic decals for the gate. You do not have to define alternates. Double-click the CAE decal to edit or select the CAE decal entry and choose Edit. Type an alternate decal name in the text box or choose the "..." Browse button to search for a decal from a library. When you choose Browse, the Assign Decal to Gate dialog box appears. This dialog box works identically to the PCB Decals tab.

Edit

Edits an entry in a multicolumn list box column. Select the entry you want to edit and choose Edit. You can also double-click on the entry to edit it.

Add Gate

Adds a gate to the part. The gate appears in the Gates multicolumn list box. The first gate is automatically designated A, the second gate B, and so forth. When you select a gate, the Pins multicolumn list box shows the pin information for the selected gate.

Delete Gate

Removes the selected gate from the Gates multicolumn list box. The gates following the removed gate are automatically renamed in sequential order.

Unused Pins

Lists all of the unused pins for the selected gate. Add pins to the gate by selecting them from the Unused Pins list and choosing Add. You can select multiple unused pins.

Add Pin

Adds the selected pin(s) from the Unused Pins list to the selected gate. You can add multiple pins at one time.

Remove Pin

Removes the selected pin(s) from the selected gate. The pin returns to the Unused Pins list. You can remove multiple pins at one time.

Pins Multicolumn List Box

Lists the pins defined for the selected gate.

Sequence	Shows the mapping of the terminal number sequence to the pin and automatically maps the sequence to alternates. In other words it shows how pins appear on the gate decal; therefore, in Gate A, sequence number 1 could be pin 1, but in Gate B, sequence number 1 would be pin 4. This option is mainly beneficial for PowerLogic users.
Pin	Lists the pins assigned to the selected gate.
Type	Assigns the pin type; for example, Source or Load. Double-click the Type entry to edit, or select the Type entry and choose Edit. Choose the Type you want to assign from the drop-down list box.
Name	Assigns a name to the pin for documentation purposes. A pin name is not required. Double-click the Name entry to edit, or select the Name entry and choose Edit. Type a name for the pin in the text box.

Swap	Assigns a Swap ID to a pin. If it is valid to swap this pin with another pin in the gate, assign a Swap ID to the pin. You can swap pins in the same gate with the same Swap ID. Double-click the Swap entry to edit, or select the Swap entry and choose Edit. Type a Swap ID in the text box.

Up and Down

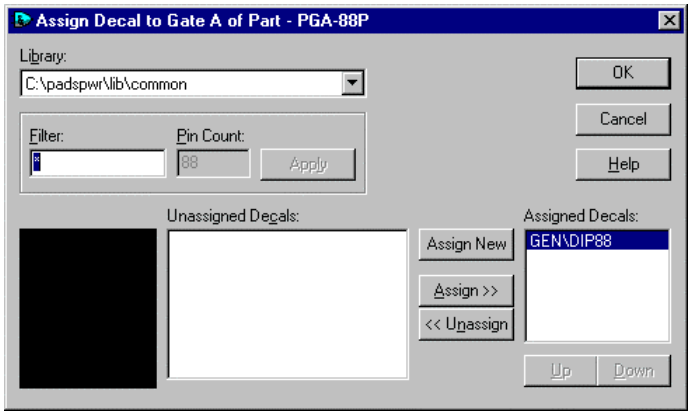
Changes the sequence of pins in the decal. This changes the order of pins; this does not effect the sequence. For example, if sequence number 1 is pin 1 and you reverse the order of pin 1 and pin 2, sequence number 1 becomes pin 2.

Assign Decal to Gate Dialog Box

This dialog box appears when you assign an alternate decal to a gate. This dialog box works identically to the PCB Decals tab except that it deals with Logic decals instead of PCB decals.

Alternates for the selected gate appear in the Assigned Decals list box in the same order as you entered them in the Gates list box. If necessary, you can reorder the alternates using Up and Down in this dialog box.

The Assign New button applies to Gate decals instead of PCB decals.



Library

Lists all libraries. Choose the library from which you want to assign decals. The available decals from the selected library appear in the Unassigned Decals list box. The list changes according to the library you select.

Filter

Limits the items that appear in the Unassigned Decals list box. Type the prefix you want to filter decals by, followed by an asterisk; for example, type **DIP*** to view all DIP decals. Type * to view all decals.

Choose Apply to see the results of your search in the Unassigned Decals list box.

Pin Count

Contains the number of pins in the decal.

Preview Area

Displays the decal that you select in the Unassigned Decals list box.

Unassigned Decals

Lists all decals from the selected library that you can assign to the part.

Assigned Decals

Lists all decals assigned to the part. You can assign up to four decals. Assigned decals must have the same number of pins. The first decal is the default decal. Use the Up and Down buttons to reorder the assigned decals.

Assign New

Assigns a decal name of a nonexistent decal in the library. This feature is very beneficial for schematic designers. You can specify a decal that doesn't exist in your library, but may exist in the PCB designers library.

When you choose Assign New, the Assign New PCB Decal dialog box appears. Type a name and choose OK. The name is added to the end of the Assigned Decals list. To unassign the new decal name, choose Unassign.

Assign

Assigns decals. To assign decals to the part:

1. Select one or more decal names from the Unassigned Decals list box.
2. Choose Assign. The decal moves to the Assigned Decals list box; the decal is assigned to the part.

Note: As a shortcut, double-click on a decal name to assign it to the part.

You can assign up to four decals. All assigned decals must have the same number of pins.

Unassign

Removes decals from the Assigned Decals list box and returns them to the Unassigned Decals list box. To delete a decal assignment:

1. Select a decal name from the Assigned Decals list box.
2. Choose Unassign. The decal moves to the Unassigned Decals list box; the decal is not assigned to the part.

Up and Down

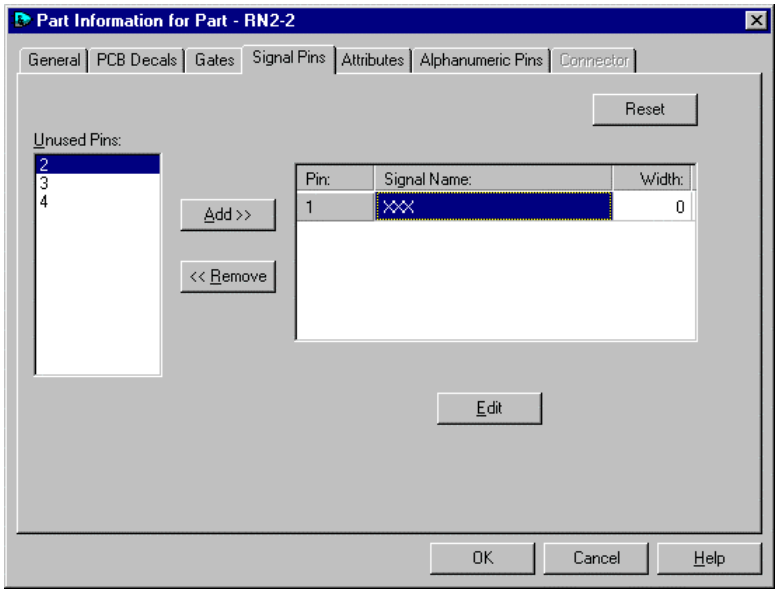
Changes the order of assigned decals in the Assigned Decals list box. The first decal is the default decal. Use Up and Down to move the decal you want to use as the default decal to the top of the Assigned Decals list box.

Reset

Returns the options on the current tab to their original settings; the settings that appeared when you first opened the tab. Reset only resets the current tab.

Signal Pins Tab

Use the Signal Pins tab in the Part Information dialog box to assign signal names to pins, such as ground and power pins. For the parts shipped with PowerLogic, the standard ground signal name is GND, the standard power signal name is +5V. You should follow this convention for your part types.



To assign a signal name to a previously unassigned pin:

1. Select the pin in the Unused Pins list box.
2. Choose Add. The pin moves to the Pin: Signal Name list box.
3. Type the signal name in the Signal Name text box.

To modify a signal name:

1. Select the pin in the Pin: Signal Name list box.
2. Type a new signal name or Width in the appropriate text box.

Unused Pins

Lists all pins that are neither assigned to gates nor have signal name assignments.

Signal Pins Multicolumn List Box

Lists information about the signal pins assigned to the part:

Pin	Lists the signal pins assigned to the part.
Signal Name	Lists the signal name for the pins. When editing the Signal Name, you can assign signal names to newly added pins or change the signal name for an existing pin assignment.
Width	Lists the signal width. The default signal width for pins is based on the rules that you specify in the Design Rules dialog box; however, for backward compatibility with PADS-Perform and PADS-Work, you can assign a default signal width to pins and store it in the library.

Double-click on the entry to edit, or select the entry and choose Edit. You can then type the new value in the text box.

Add

Adds a pin to the part. Select the pin to add from the Unused Pin list box and choose Add. The pin is added to the Signal Pins multicolumn list box. You can assign a signal name after adding the pin.

Remove

Removes the selected signal name assignments from the Signal Pins multicolumn list box.

Attributes Tab

Use the Attributes tab in the Part Information dialog box to add an attribute to a part. See the "Working with Attributes" chapter for information on adding attributes to the libraries or at the schematic level.

To add an attribute to a part:

1. Choose Add.
2. Type an attribute name in the Attribute column.
3. Type an attribute value in the Value column.

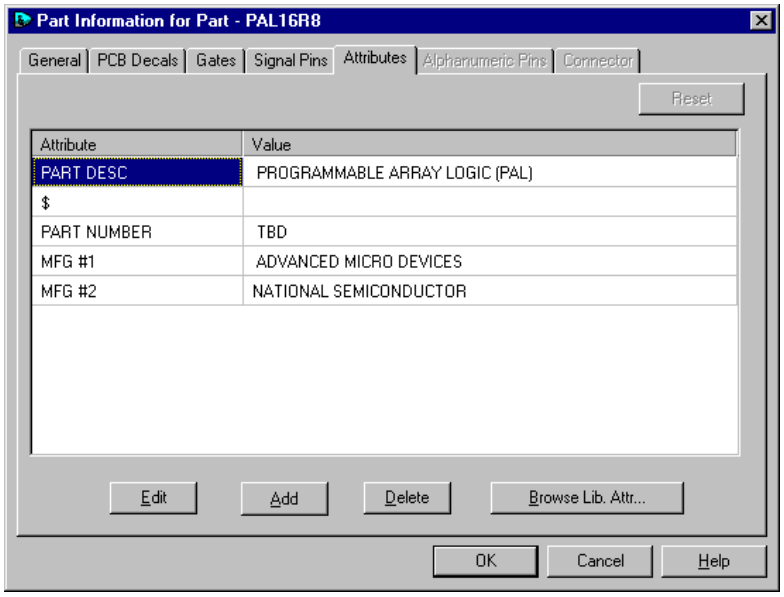
To modify an attribute:

1. Double-click on an attribute from the Attribute multicolumn list box, or select the attribute and choose Edit.
2. Type a new name or value in the appropriate text box.

To delete an attribute from the part:

1. Select the attribute.
2. Choose Delete.

Note: Modifying attributes in the Attributes tab modifies the attribute only in the part being edited. Use the Manage Schematic Attributes dialog box and the Library Manager dialog box to manage attributes design-wide or in all libraries.



Attribute multicolumn list box

Lists assigned attributes and their corresponding values.

Attribute — Lists the attributes assigned to the part.

Value — Lists the value of the attributes assigned to the part.

To edit an attribute or value, double-click on the entry to edit or select the entry and choose Edit. Type the new attribute or value in the text box.

Add

Adds a new attribute to the part you are editing or creating. The attribute is added at the end of the current list. You must assign a name to new attributes.

Delete

Deletes the selected attribute from the Attribute multicolumn list box.

Browse Lib. Attr

Opens the Browse Library Attributes dialog box that lists all attribute names in all libraries. Choose an attribute to add from this list. See the "Browse Library Attributes Dialog Box" topic in the "Library Operations" chapter for more information.

Alphanumeric Pins Tab

Use the Alphanumeric Pins tab in the Part Information dialog box to assign alphanumeric names to pins. Alphanumeric pin names are optional; often the pin number is sufficient as a pin name.

To assign alphanumeric names, you must first turn on the Alphanumeric Pins for Part check box. When this check box is off, the rest of the tab grays.

The pin number displayed in the list box is the CAE decal pin number. This allows you to associate the decal pin with the alphanumeric pin name. Initially, the program assigns the pin number as the pin name.

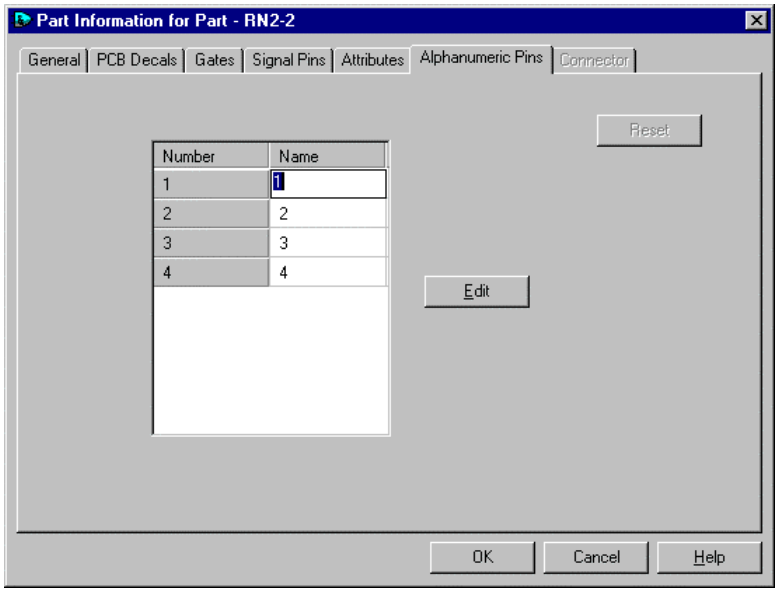
Alphanumeric pin names can be a maximum of 4 characters. All alphanumeric characters are accepted. You cannot use special characters, such as exclamation marks !, asterisks @, and dollar signs $.

If alphanumeric pin names are specified in the electrical description, they are substituted for the part type pin numbers in the schematic and the netlist. For example, in place of pin number 1, the first alphanumeric pin name will appear, in place of pin number 2, the second alphanumeric pin name listed will appear, and so forth.

You must define all terminals with an alphanumeric name when non-numeric pin names are enabled for a part type. If you only need a few pins defined as alphanumeric, then define the non-alphanumeric pins as the terminal number. For example, IN OUT 3 4 5 GND.

To change the pin name:

1. Double-click on the pin you want to change in the Alphanumeric Pins multicolumn list box, or select the pin and choose Edit.
2. Type a new name in the Name column.



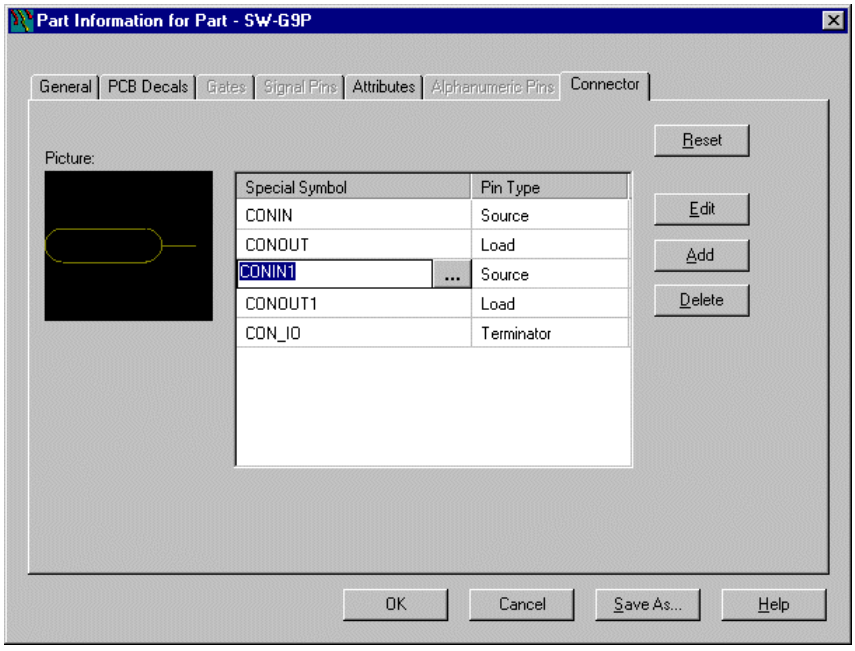
Alphanumeric Pins Multicolumn List Box

Lists assigned pins by number and any associated name you define.

Number	Lists the decal pin number. You can associate the decal pin with the alphanumeric pin name. Initially, PowerPCB assigns the pin number as the pin name. You cannot edit the pin number.
Name	Lists the name associated with the decal pin number. To edit the name, double-click on the Name entry or select the entry and choose Edit. Type a new name in the text box.

Connector Tab

This tab is only enabled when you open an existing connector or are creating a new connector. Use this tab to define the alternate Logic decals to display in a schematic. Decals are referred to as Special Symbols. You can associate a logical Pin Type with each alternate so that you can have a graphical indication of the connector pin function in the schematic. Most users like to use a different symbol, or decal, to distinguish between input (Source) and output (Load) pins. You may define multiple symbols for each of the ten different pin types.



Connector Multicolumn List Box

Use this multicolumn list box to define Special Symbols, or connector decals, for different pin types in the part.

Special Symbol	Lists the Special Symbols, or decals, defined as connectors for the current part. To edit the name of the Special Symbol, double-click on the entry or select the entry and choose Edit. Type a new symbol name or choose the "..." Browse button to search for a symbol from a library. When you choose Browse, the Browse for Special Symbols dialog box appears.
Pin Type	Lists the pin type associated with the selected Special Symbol. To edit the pin type, double-click on the entry or select the entry and choose Edit. Choose one of the 10 pin types from the drop-down list box.

Add Special Symbol

Adds a new entry in which you can define a new Special Symbol.

1. Choose Add.
2. Double-click in the new Special Symbol entry or select the new entry and choose Edit.
3. Type a Special Symbol or choose one from a library using the "..." Browse button. The Browse button opens the Browse for Special Symbols dialog box.
4. Double-click on the new Pin Type entry or select the new entry and choose Edit.
5. Choose a Pin Type from the Pin Type drop-down list box.

Delete Special Symbol

Deletes the selected Special Symbol from the multicolumn list box.

Creating New Parts from Existing Parts

A fastest way to make a new part is to alter a copy of an existing part. Modifying an existing CAE decal without first copying it modifies all parts that use that decal. To prevent this:

- Verify the CAE decal assigned to the part of which you want to copy references.
- Modify that CAE decal and save it with a new name.

To make a copy of the CAE Decal:

1. Choose Edit/Part Editor.
2. Choose File/Open. The Select Type of Editing Item dialog box appears. See the "Select Type of Editing Item" topic in this chapter for more information.
3. Choose the CAE Decal button.

4. Choose OK. The Get Gate Decal from Library dialog box appears.
5. Set the filter, and then highlight the part in the Decal list box.
6. Choose OK. You enter the Part Editor and the decal appears.
7. Modify the decal as required.
8. Choose File/Save As. The Save Item to Library dialog box appears.
9. Type a name for the new CAE decal and choose a library directory.
10. Choose OK
11. Choose File/Exit Part Editor.

To make a copy of the part, use the steps for copying a CAE decal (above), except choose Part Type instead of CAE Decal in Step 3. After you change the CAE decal and the electrical information, save it as a new part.

Select Type of Editing Item

Use this dialog box to choose the type of library item to create or modify.

Part Type

Creates or modifies an existing part. See the "Get Part Type from Library" topic in this chapter for more information.

Connector

Creates or modifies a connector. See the "Creating a New Connector" topic in this chapter for more information.

CAE Decal

Creates or modifies new CAE decal. See the "Get Gate Decal from Library" topic in this chapter for more information.

Pin Decal

Creates or modifies a pin decal, the information and appearance of a terminal pin. A number of different pin decals are provided.

Off Page

Modifies off-page reference symbols. PowerLogic allows only one part definition in the library for off-page reference symbols. This option grays when you choose File/New. Modify existing symbols and/or add new symbols. See the "Special Symbols" topic in this chapter for more information.

Power

Modifies existing off-page reference symbols. PowerLogic allows only one part definition in the library for power symbols. This option grays when you choose File/New. Modify existing symbols and/or add new symbols. See the "Special Symbols" topic in this chapter for more information.

Ground

Modifies an existing off-page reference symbol. PowerLogic allows only one part definition in the library for ground symbols, so this option grays when you choose File/New from the menu. You can modify the existing symbols and/or add new symbols. See the "Special Symbols" topic in this chapter for more information.

Creating a New Connector

Connectors differ from other parts because a connector is typically represented as a number of individual pins, rather than a single complete part. When you use a connector on a schematic, the pin number rather than the reference designation increments, for example, P1-1, P1-2, P1-3, instead of P1, P2, P3.

Also, you can assign several CAE Decals to a connector for left and right side locations on the schematic and for alternate decals. The connectors supplied with PowerLogic use the prefix CON in their name. You can open an existing connector to see how connectors are constructed.

To create a new connector:

1. Choose Tools/Part Editor.
2. Choose Edit/Electrical, or choose the Edit Electrical icon from the toolbar. The Part Information dialog box appears.
3. Turn the Connector check box on. This modifies the Part Information dialog box for building a connector. See the "Connector Tab" topic in this chapter for more information.
4. Use Pin Count to indicate the number of pins in the connector. Type the number of pins for the entire connector, not the individual connector symbol. Do not include mounting holes in the total.
5. Optionally, choose the Decals icon from the Part Information dialog box to assign the PCB decal. The pin count for the assigned PCB decal must be equal to or greater than the electrical description.
6. Choose the Pin Decals icon from the Part Information dialog box. The Assign Alternatives for Connectors dialog box appears.

7. Assign decals for the connector. See the "Assign Alternatives for Connector" topic in this chapter for more information.
8. Choose OK.

9. Choose the Attribute icon from the Part Information dialog box to assign attribute information.
10. Choose File/Save As.
11. Type a name and library directory location.
12. Choose OK.
13. Choose File/Exit Part Editor.

Assign Alternatives for Connector

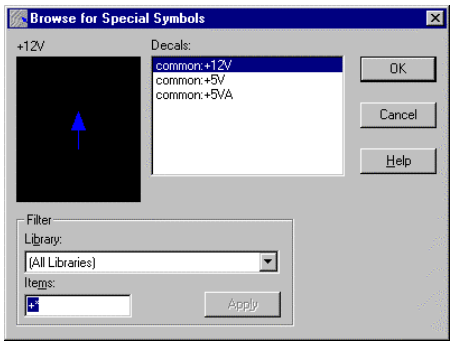
Use the Assign Alternatives for Connectors dialog box to assign pin decals for a connector. To add a new CAE decal to a connector:

1. Choose the Pin Decals icon from the Part Information dialog box.
2. Choose Add. Type a name for the decal in newly created text box in the Special Symbol column, or choose the icon to the right of the new text box. The Browse for Special Symbol dialog box appears.
3. Set the filter to search for decals beginning with CON, for example, CON*.
4. Choose a decal.
5. Choose OK.
6. Select the text box under Pin Type.
7. Choose Edit.
8. Choose a pin type from the drop-down list box.
9. Choose OK.
10. Repeat steps 2 through 9 for any other decals. Specify a connector for the left and right side, for example, use CONIN and CONOUT.
11. Choose OK.

Browse for Special Symbols

Use the Browse for Special Symbol dialog box to access the special symbols provided with the PowerLogic library. Special symbols create off-page reference symbols, power symbols, ground symbols, and connectors. See the "Special Symbols" topic in this chapter for more information.

1. Set the filter to search for specific decals, for example, CON* to search for connector symbols.
2. Choose a decal from the list.
3. Choose OK.



Decals

Displays the decals that PowerLogic matched to the search filter settings. Choose the decal you want to use from this list. The selected decal appears in the area to the left of the list box.

Filter

Searches the selected library for a specific part or item name or for names that match a wildcard text string. See the "Preface" for more information.

Creating a New Pin Decal

Pin Decals represent the terminal pins on a part. You can make a copy of an existing pin decal and modify it or you can create a new pin decal. To create a new pin decal from an existing pin decal:

1. Choose Tools/Part Editor to enter the Part Editor.
2. Choose File/Open. The Select Type of Editing Item dialog box appears.
3. Choose Pin Decal.
4. Choose OK. The Pin Decal Browse dialog box appears.
5. Choose an existing pin decal from the Pins area. The highlighted decal appears in the Picture area.
6. Choose OK.
7. Modify the graphics for the decal and position the text strings as required.
8. Choose File/Save As. The Save Item to Library dialog box appears.
9. Type a name and library location for the new pin decal.
10. Choose OK.

When you create a new Pin Decal, PowerLogic automatically creates the necessary text strings for the associated pin information. You only need to specify the 2D lines that represent the terminal pin, then reposition the text strings.

To create a new pin decal.

1. Choose Tools/Part Editor to enter the Part Editor.
2. Choose File/New. The Select Type of Editing Item dialog box appears. See the "Select Type of Editing Item" topic in this chapter for more information.
3. Choose Pin Decal.
4. Choose OK.

   Four text entries appear in the work area:

#E	Placeholder for the pin number.
PNAME	Placeholder for the pin name.
NETNAME	Placeholder for the net name.
#0:TYP=U SWP=0	Placeholders for the sequence number, pin type and swap class.

5. Choose the Create 2D Line icon from the Drafting toolbox to create the pin decal.
6. Reposition the text strings.
7. Choose File/Save. The Save Item to Library dialog box appears.
8. Type a name and library location for the new pin decal.
9. Choose OK.

Managing Pin Decals

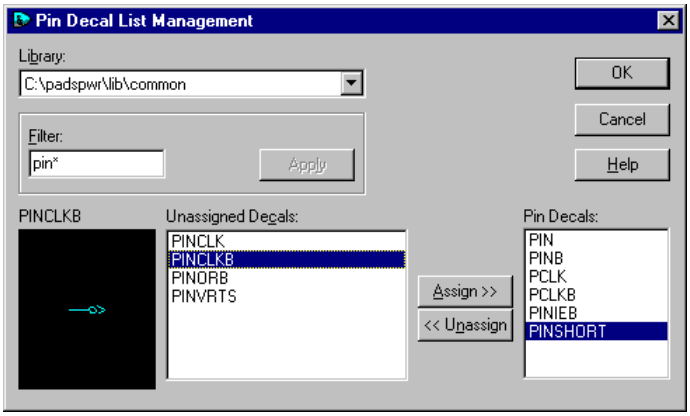
Use the Pin Decal List Management dialog box to control which decals are displayed in the Pin Decal List. The pin decal list contains the decals that can be used as terminal graphics when creating gate or part decals.

Choose Setup/Pin List Manager to open this dialog box.

> Note: This dialog box is only available when you are in the Part Editor.

Pin Decal List Management Dialog Box

Use the Pin Decal List Management dialog box to manage the list of available pin decals.



Filter

Searches the chosen library (or libraries) for a specific item name, or names that match a wildcard text string.

Use the Library drop-down list to choose specific library directories or the All Libraries setting.

Type the prefix that you want to match up with library pin decal names, followed by an asterisk, in the Items text box (for example, type PIN* to view all pin decals that begin with PIN). Type * to view all pin decals in the chosen libraries.

Choose Apply to search the libraries and display the search results.

Preview Area

Displays the pin decal highlighted in the Unassigned Decals area or the Pin Decals area.

Unassigned Decals

Lists the available pin decals in the selected library or all libraries.

Assign

Moves a decal from the Unassigned Decals list box to the Pin Decals list. Choose a decal, then choose Assign.

Unassign

Moves a decal from the Pin Decals list box to the Unassigned Decals list box. Choose a decal, then choose Unassign.

Pin Decals

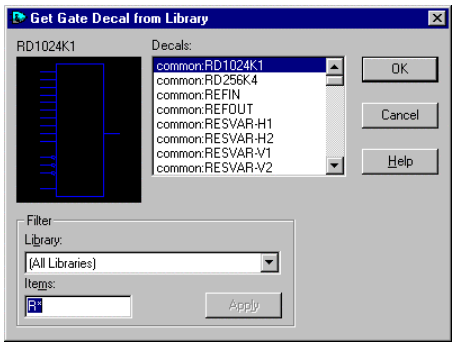
Lists the pin decals that are displayed in the dialog boxes that access pin decals, for example, the Pin Decal Browse dialog box. You can display up to 100 pin decals.

Get Gate Decal from Library

Use the Get Gate Decal from Library dialog box to open an existing CAE Decal in the Part Editor.

To open this dialog box:

1. Choose Tools/Part Editor.
2. Choose File/Open. The Select Type of Editing Item dialog box appears.
3. Choose CAE Decal.
4. Choose OK.



Decals

Displays the decals that match to the search filter. Choose the decal you want to use. The selected decal appears in the area to the left of the list box.

Filter

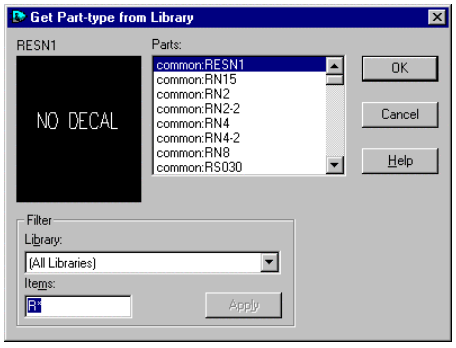
Searches the selected library for a specific part or item name or for names that match a wildcard text string. See the "Preface" for more information.

Get Part Type from Library

Use the Get Part Type from Library dialog box to open an existing part in the Part Editor.

To open this dialog box:

1. Choose Tools/Part Editor.
2. Choose File/Open. The Select Type of Editing Item dialog box appears.
3. Choose Part Type.
4. Choose OK.



Decals

Displays the decals that PowerLogic matched to the search filter settings. Choose the decal you want to use from this list. The selected decal appears in the area to the left of the list box.

Filter

Searches the selected library for a specific part or item name or for names that match a wildcard text string. See the "Preface" for more information.

Part Editor Drafting Toolbox

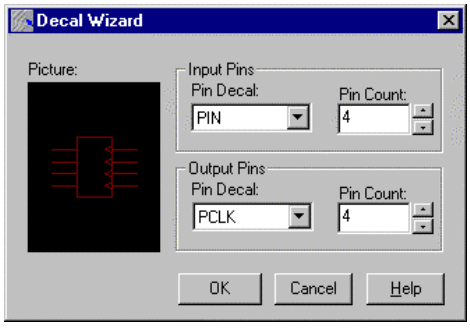
The Part Editor's Drafting toolbox uses the same commands and icons as the Schematic Editor's Drafting toolbox, but has an additional icon, the Decal Wizard, for creating new parts. See the "Drafting Toolbox" topic in the "Drafting Operations" chapter for more information.

Decal Wizard

To open the Decal Wizard and create a new part, choose the Decal Wizard icon from the Drafting toolbox in the Part Editor. Use the Decal Wizard dialog box to automatically create a new CAE decal. The icon that opens the Decal Wizard appears at the end of the Drafting toolbox when you are in the Part Editor. You must be in the Part Editor and creating or modifying gate information to use this feature.

In this wizard you can specify the pin decal to use and the number of pins for the left and right side of the decal. It then creates a rectangle of the correct size, adds the pins, and creates place holders for the sequence number, pin number, pin type, and swap information.

Use the Terminal toolbox to modify the basic information created by the decal wizard. See the "Terminal Toolbox" topic in this chapter for more information. You can add additional terminals, change the pin decal, or change the pin sequence number. The other options in the Terminal toolbox gray when you create a CAE Decal.



Input Pins Area

Selects the pin decals for the left, or input, side of the part. Type the number of pins in the Pin Count text box.

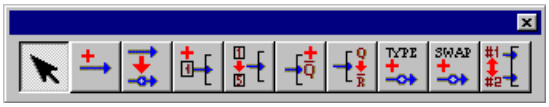
Output Pins Area

Selects the pin decals for the right, or output, side of the part. Type the number of pins in the Pin Count text box.

Terminal Toolbox

Use the Terminal toolbox to add pin terminals to the decal and to assign pin names, pin numbers, pin types, and pin swapping information. Pin numbers correspond to component pads or pins in the PCB decal when the part is used in a PCB design database.

Note: You must be in the Part Editor to view the Terminal Toolbox icon.



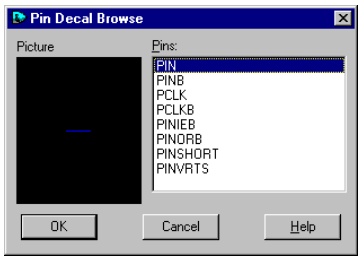
	Add Terminal	Adds a connection pin to the part. A terminal consists of a pin decal and a series of text strings that define the terminal's number, swap data, etc. See the "Adding Terminals" topic in this chapter for more information.
	Change Pin Decal	Selects a different pin decal for an existing terminal. See the "Changing Pin Decals" topic in this chapter for more information.
	Set Pin Number	Assigns pin numbers to several pins. The pin number is incremented each time you select a pin. See the "Setting Pin Numbers" topic in this chapter for more information.
	Change Pin Number	Changes the number assigned to a pin. This command does not check for duplicate numbering. See the "Changing Pin Numbers" topic in this chapter for more information.
	Set Pin Name	Adds or changes the name for several pins. Pin names are used to label the function of a pin. This command does not check for duplicate naming conventions. See the "Setting Pin Names" topic in this chapter for more information.
	Change Pin Name	Changes the name assigned to a single pin. Pin names label the function of a pin. This command does not check for duplicate names. See the "Changing Pin Names" topic in this chapter for more information.
	Set Pin Type	Changes the type of pin the terminal represents, for example, load, source, etc. See the "Setting Pin Types" topic in this chapter for more information.

	Set Pin Swap	Defines a swap class for a terminal. Swapping information is used in the PCB Design system for length minimization and routing optimization. See the "Setting Pin Swaps" topic in this chapter for more information.
	Change Sequence Number	Changes the sequence for pins in the part. The pin sequence number is used to set a corresponding number for the pins in alternate CAE decals. See the "Changing Sequence Numbers" topic in this chapter for more information.

Adding Terminals

Use Add Terminal from the Terminal toolbox to add a connection pin to the part. A terminal consists of a pin decal and a series of text strings that define the terminal's number, swap data, etc. To add a new terminal:

1. Choose the Add Terminal icon from the Terminal toolbox. The Pin Decal Browse dialog box appears.



   The Pins area lists all terminal pins. The selected pin appears in the Picture area.

2. Choose a decal, then choose OK. The pin decal attaches to the cursor.
3. Move the cursor and click the left mouse button to anchor the pin.

   Before placing the terminal, you can use the pop-up menu to modify the orientation of the terminal and modify terminal information, including the pin number, pin name, pin type, and swap class.

   After you place a terminal, a new, identical terminal attaches to the cursor for placement. Pin numbers or pin names automatically increment to the next number and name for the new terminal. Pin type and swap class values copy to the new terminal.

   Optionally, you can add multiple copies of the terminal after the initial terminal location is indicated. Choose Step and Repeat from the pop-up menu. See the "Step and Repeat" topic in the "Design Operations" chapter for more information.

4. Choose Cancel from the pop-up menu when you finish placing terminals.

Changing Pin Decals

Use Change Pin Decal from the Terminal toolbox to select a different pin decal for an existing terminal.

1. Choose the Change Pin Decal icon from the Terminal toolbox. The Pin Decal Browse dialog box appears.
2. Choose a new pin decal from the list.
3. Choose OK.

Select the terminals to change.

Setting Pin Numbers

Use Set Pin Number from the Terminal toolbox to assign pin numbers to several pins. The pin number is incremented each time you select a pin.

1. Choose the Set Pin Number icon from the Terminal toolbox.
2. Type the starting number in the information window.
3. Choose OK.
4. Select each pin to assign a number to in ascending sequence. To skip a number, select the same pin until the correct pin number appears, then continue selecting other pins.
5. Choose Cancel from the pop-up menu when you finish setting numbers.

If you turned the Alpha Pins check box on in the Part Information dialog box, you can type an alphanumeric character in the information window to assign letters to the pins.

Use the Change Pin Number icon from the Terminal toolbox to change individual pin numbers.

Changing Pin Numbers

Use Change Pin Number from the Terminal toolbox to change the number assigned to a pin. This command does not check for duplicate numbering.

1. Choose the Change Pin Number icon from Terminal toolbox.
2. Select the pin requiring a new number.
3. Type the new pin number in the information window.
4. Choose OK.

Use the Set Pin Number icon from the Terminal toolbox to assign numbers to several pins in an ascending sequence.

Setting Pin Names

Use Set Pin Name from the Terminal toolbox to add or to change the name for several pins. Use pin names to label the function of a pin, for example, CLK, DATA0, etc. The suffix of the name is incremented each time you select a pin. This command does not check for duplicate naming conventions.

Precede the text with a backslash (\) to add a bar over text characters.

1. Choose the Set Pin Name icon from the Terminal toolbox.
2. Type the pin name in the information window.
3. Choose OK.
4. Select each pin to which to assign a name. To skip a name, select the same pin until the correct pin name appears, then continue selecting other pins.
5. Choose Cancel from the pop-up menu when you finish.

Use the Change Pin Name icon from the Terminal toolbox to change the name of a single pin.

Changing Pin Names

Use Change Pin Name from the Terminal toolbox to change the name assigned to a single pin. Use pin names to label the function of a pin, for example, CLK, DATA0, etc. This command does not check for duplicate naming conventions.

Precede the text with a backslash (\) to add a bar over text characters.

1. Choose the Change Pin Name icon from the Terminal toolbox.
2. Select the pin requiring a new name.
3. Type the new name in the information window.
4. Choose OK.

Use the Set Pin Name icon from the Terminal toolbox to assign names to several pins in an ascending sequence.

Setting Pin Types

Use Set Pin Type from the Terminal toolbox to change the type of pin the terminal represents, for example, load, source, etc.

1. Choose the Set Pin Type icon from the Terminal toolbox. The Pin Type information window appears.
2. Choose a pin type from the drop-down list box.
3. Select the pins to which to apply the pin type information. The text string associated with the pin reflects the new pin type. The characters that represent the pin type are listed below.

4. Choose a new pin type if necessary; choose additional pins.

   Characters used to represent pin types:

B	Bidirectional Pin
C	Open Collector Pin
G	Ground Pin
L	Load Pin Or-tieable Source Pin
P	Power Pin
S	Source Pin
T	Tristate Pin
U	Undefined
Z	Terminator Pin (For ECL rules)

Setting Pin Swaps

Use Set Pin Swap from the Terminal toolbox to define a swap class for a terminal. Swapping information is used in the PCB design system for length minimization and routing optimization. The swap class is assigned by a numeric value.

The number for classes are 0, if the pin is not to be swapped under any circumstances, and 1 through 99. A pin can be swapped with any other pin within the gate which has the same swap class number.

1. Choose the Set Pin Swap icon from the Terminal toolbox.
2. Type the character that identifies the swap class in the information window.
3. Choose OK.
4. Select each pin to assign to this swap class.
5. Choose Cancel from the pop-up menu when you finish setting pin swaps.

Changing Sequence Numbers

Use Change Sequence Number from the Terminal toolbox to change the sequence for pins in the part. The pin sequence number sets a corresponding number for the pins in alternate CAE decals. When you assign pin numbers using Set Pin Number or change a pin number using Change Pin Number, the same number is assigned to any alternate decals.

1. Choose the Change Sequence Number icon from the Terminal toolbox.
2. Select the pin requiring a new sequence number.
3. Type the new sequence number in the information window.
4. Choose OK.

Special Symbols

Special symbols in PowerLogic include power symbols, ground symbols, and off-page reference symbols. PowerLogic allows only one part definition in the library for special symbols. For this reason, the options for these types gray when you choose File/New. You can modify the existing symbols and/or add new symbols. See the "Managing Special Symbols in the Library" topic in this chapter for more information.

Creating New Special Symbols

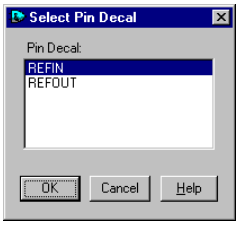
Special symbols in PowerLogic include power symbols, ground symbols, and off-page reference symbols. PowerLogic allows only one part definition in the library for special symbols. For this reason, the options for these types gray when you choose File/New.

To create a new special symbol:

1. Choose Tools/Part Editor.
2. Choose File/Open. The Select Type of Editing Item dialog box appears.
3. Choose the type of symbol to add: Off-page, Power, or Ground.
4. Choose OK. The current symbols for the selected type appear.
5. Choose Edit/Electrical, or choose the Edit Electrical icon from the toolbar. The Assign Alternatives dialog box for the type of special symbol appears.
6. Choose Add. A new entry is created.
7. Type a name for the new symbol and enter the appropriate information.
8. Choose OK. An error message appears since you haven't created the symbol yet. Answer No to the prompt.
9. Choose Edit/Gate, or choose the Edit Gate Decal icon from the toolbar.
10. Choose Extended Selection from the pop-up menu. The Select Pin Decal dialog box appears.
11. Choose the new symbol name. Answer OK to create the new symbol.
12. Use the Create 2D Line icon from the Drafting toolbox to create the symbol.
13. Reposition the text strings as required.
14. Choose File/Complete. The new symbol and existing symbols appear.
15. Choose File/Save.
16. Choose File/Exit Part Editor.

Select Pin Decal

Use the Select Pin Decal dialog box to access and create graphics for a new special symbol. This dialog box appears when you choose the Extended Selection command from the pop-up menu to edit a gate for a special symbol.



To create graphics for a new symbol name:

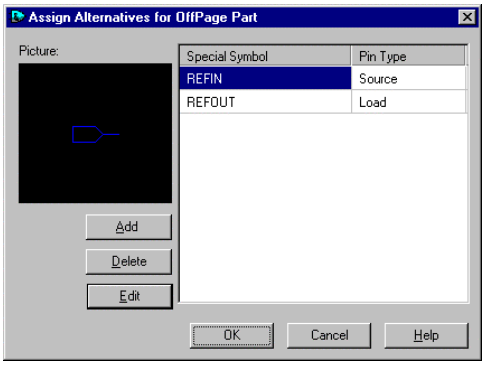
1. Choose the new alternate symbol name.
2. Choose OK. A Create New Decal information window appears.
3. Choose OK. You enter the special part editor and the appropriate text strings are added.

You can also modify one of the existing parts by choosing it from the list.

Assign Alternatives for Off-page Part

Use the Assign Alternatives for Off-page Part dialog box to assign or create additional off-page reference symbols. To open this dialog box:

1. Choose Tools/Part Editor.
2. Choose File/Open. The Select Type of Editing Item dialog box appears.
3. Choose Off-page.
4. Choose OK. The current symbols appear.
5. Choose Edit/Electrical, or choose the Edit Electrical icon from the toolbar.



To add a new off-page reference symbol:

1. Choose Add. This creates a new entry for the symbol.
2. Type a name for the new symbol. Note the existing naming conventions used.

   If the symbol already exists:

   a. In the Special Symbol column, choose the icon to the right of the newly created text box. The Browse for Special Symbol dialog box appears.

   b. Set the filter for an existing symbol, for example REF*.

   c. Choose a symbol decal.

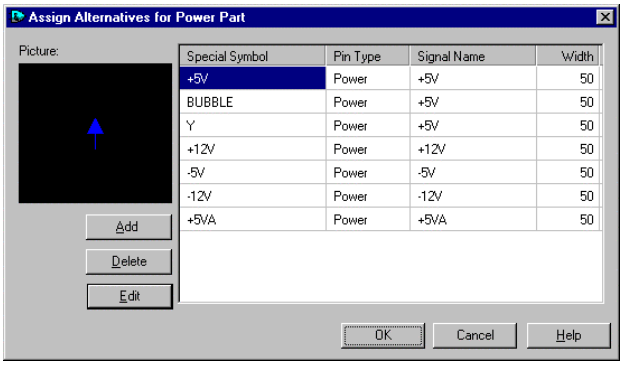
   d. Choose OK.
3. Select the text box under Pin Type.
4. Choose Edit.
5. Choose a pin type from the drop-down list box. For left side off-page references, choose Source; for right side off-page references, choose Load.
6. Choose OK.

Assign Alternatives for Power Part

Use the Assign Alternatives for Power Part dialog box to assign or create additional power symbols.

To open this dialog box:

1. Choose Tools/Part Editor.
2. Choose File/Open. The Select Type of Editing Item dialog box appears.
3. Choose Power.
4. Choose OK. The current symbols appear.
5. Choose Edit/Electrical, or choose the Edit Electrical icon from the toolbar.



To add a new power symbol:

1. Choose Add. This creates a new entry for the symbol.
2. Type a name for the new symbol. Note existing naming conventions.

   If the symbol already exists:

   a. In the Special Symbol column, choose the icon to the right of the newly created text box. The Browse for Special Symbol dialog box appears.

   b. Set the filter for an existing symbol, for example +* or -*.

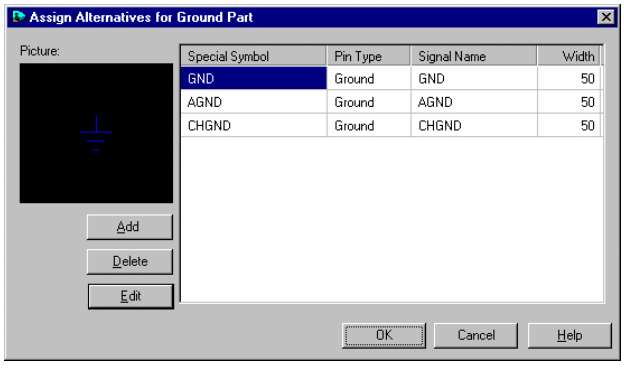
   c. Choose a symbol decal.

   d. Choose OK.
3. Select the text box under Pin Type.
4. Choose Edit.
5. Choose a pin type from the drop-down list box. This is normally set to Power.
6. Select and edit the text boxes for the Signal Name and Width.
7. Choose OK.

Assign Alternatives for Ground Part

Use the Assign Alternatives for Ground Part dialog box to assign or create additional ground symbols. To open this dialog box:

1. Choose Tools/Part Editor.
2. Choose File/Open. The Select Type of Editing Item dialog box appears.
3. Choose Ground.
4. Choose OK. The current symbols appear.
5. Choose Edit/Electrical, or choose the Edit Electrical icon from the toolbar.



To add a new ground symbol:

1. Choose Add. This creates a new entry for the symbol.
2. Type a name for the new symbol. Note the existing naming conventions used.

   If the symbol already exists:

   a. In the Special Symbol column, choose the icon to the right of the newly created text box. The Browse for Special Symbol dialog box appears.

   b. Set the filter for an existing symbol, for example GND*.

   c. Choose a symbol decal.

   d. Choose OK.
3. Select the text box under Pin Type.
4. Choose Edit.
5. Choose a pin type from the drop-down list box. This is normally set to Ground.
6. Select and edit the text boxes for the Signal Name and Width.
7. Choose OK.

Creating New Special Symbols from Existing Symbols

Special symbols in PowerLogic include power symbols, ground symbols, and off-page reference symbols. PowerLogic allows only one part definition in the library for off-page reference symbols, so this option grays when you choose File/New from the menu. You can modify the existing symbols and/or add new symbols.

See the "Creating New Special Symbols" topic in this chapter for information on creating and assigning a new special symbol.

Use the Library Manager to make a copy of an existing special symbol:

1. Choose File/Library.
2. Choose the CAE icon from the Library Manager dialog box.

3. Set the filter for the symbol to search for, for example +* for power symbols. See the "Special Symbol Naming Conventions" topic for more information.
4. Double-click on the symbol in the Decal area.
5. Choose Copy. The Save Item to Library dialog box appears.
6. Type a name and director location for the new special symbol.
7. Choose OK.

To modify the new special symbol while in the Library Manager:

1. Set the filter for the symbol if you saved it with a different prefix.
2. Select the symbol in the CAE Decals area.
3. Choose Edit.
4. Minimize the Library Manager dialog box or move it to one side.

5. Modify the graphics and rearrange the text strings as required.
6. Choose File/Save.

You can close the Library Manager dialog box or use it for another operation.

Special Symbol Naming Conventions

The special symbols in PowerLogic's library use specific names or a unique naming prefix. Use the chart below to locate a special symbol in the library.

Off-page Reference Symbols

REFIN	Off-page reference for left side.
REFOUT	Off-page reference for right side.

Power Symbols

+	+5 volt and +12 volt symbols, arrow pointing up. 5 volt and 12 volt symbols, arrow pointing down.
BUBBLE	+5 volt symbol with a circle.
Y	+5 volt symbol with a 'Y' shape.

Ground Symbols

GND	Standard ground.
AGND	Analog ground.
CHGND	Chassis ground.

Updating Off-page Symbols from the Library

Use Update Off-page from Library to update off-page, ground, or power symbols in the current schematic with the version in the library. Modifying an off-page reference symbol only affects the library entry of that part, not the current version the schematic.

1. Choose the symbols to update.
2. Choose OK.

Updating Pins from Library

Use Update Pins from Library to update the terminal pins in the schematic with the version in the library.

1. Choose the symbols to update. Choose Select All to update all terminal pins.
2. Choose OK.

Save Off-page Symbols to Library

Use Save Off-page to Library to update the off-page, ground, or power symbols in the library with the current version(s) in the schematic. Choose Tools/Save OffPage to Library to open this dialog box.

1. Choose the symbol to update.
2. Choose OK. The selected item is opened in the Part Editor.
3. Modify the symbols as required.
4. Choose File/Save.

Follow the prompts to replace the versions currently in the library and update the symbols in the schematic.

Managing Special Symbols in the Library

Special symbols create off-page reference, power, and ground symbols, and connectors. The default location of these special symbols is the \padspwr\lib\common library.

Reinstalling the library overwrites the \padspwr\lib\common library. If you modify a special symbol and want to keep the change, use Library/Export to transfer the symbol from the \padspwr\lib\common library to your \padspwr\lib\usr library or another library name.

Setting the Origin for a Part

Use Setup/Set Origin to specify a new origin for a CAE Decal. When you add a part in PowerLogic, the point of insertion is determined by the origin of the decal.

While creating or modifying a decal:

1. Choose Setup/Set Origin.
2. Indicate a new origin. An information window with the new X,Y coordinate appears.
3. Choose Yes to accept the new origin.

Check Part

Use Tools/Check Part to verify that a part contains all necessary information for use in a design and that no errors are present; unnumbered terminals, duplicate terminal numbers, proper gate assignments, etc. Error messages appear on the screen. PowerLogic does not allow you to use parts with errors in a design.

Chapter 8
Drafting Operations

This chapter discusses drafting commands in PowerLogic and how to use the Query/Modify mode to get information on and modify objects in PowerLogic. The topics in this chapter include:

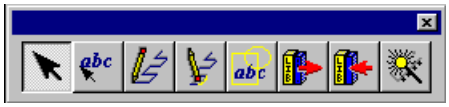
Drafting Toolbox
Query/Modify

Drafting Toolbox

The Drafting toolbox provides drafting commands to create lines, arcs, text, and all other items not generally associated with connectivity.

Use the Drafting toolbox to create and edit drafting objects. Use pop-up menus to choose the shape you want to draw or to cancel a command.

To open the Drafting toolbox, choose the Drafting Toolbox icon from the toolbar.



	Add Text	Adds a free text item to the schematic. See the "Adding Text" topic in this chapter for more information.
	Create 2D Line	Adds a 2D line item, such as a polygon, circle, rectangle, or path. Also, specifies the drafting angle for and adds arcs to polygons or paths. See the "Creating 2D Line Items" topic in this chapter for more information.
	Modify 2D Line	Modifies 2D line items using the following techniques: • Pulls an arc from a segment or corner. • Splits a segment. • Changes the line width of an item. • Fills a closed polygon. • Changes the line style to solid or dotted. See the "Modifying 2D Line Items" topic in this chapter for more information.

	Combine	Combines selected 2D line items into a complex line item. See the "Combining 2D Line Items" topic in this chapter for more information.
	Add from Library	Adds a 2D line item from a library. See the "Adding Drafting Items from a Library" topic in this chapter for more information.
	Save to Library	Save a 2D line item to a library. See the "Adding Drafting Items to a Library" topic in this chapter for more information.

Adding Text

To add *free text*, text not belonging to another object, choose the Add Text icon from the Drafting toolbox. The Add Free Text dialog box appears for you to type the text and specify other text string properties.

All alphabetic, numeric, and special characters on the keyboard are valid. The maximum text string length is 72 characters, including spaces.

To add a text entry with a bar over the characters, precede the text with the \ character. To place a bar over only a portion of text, enclose that section with the \ character. For example, typing \READ\WRITE places a bar over READ.

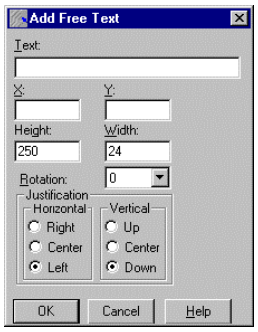
1. Choose the Add Text icon from the Drafting toolbox.
2. Type the text string. The other fields are optional.
3. Choose OK. If you did not specify the X,Y coordinate location, the text item follows the cursor movement.
4. Indicate the text item location if necessary.

Optionally, you can add multiple copies of the text after the initial text location is indicated. See the "Step and Repeat" topic in the "Design Operations" chapter for more information.

To modify a free text item:

1. Choose the Query/Modify Mode icon from the toolbar.
2. Select the text string to modify.

   Text appears at its true width if the Real Width value in the Global tab is equal to or less than the text line width.



Text

Specifies the text string that composes the text line item.

X and Y

Locates the text string at the precise X,Y coordinate location you type. If this is blank when you choose OK, the text attaches to the cursor until you indicate the location.

Height and Width

Overrides the default text height and width for this specific string with the values you specify. Height specifies the character height; type a value between 10 and 1000. Width specifies the line width of the character font; type a value between 1 and 50. To change the default height and width for free text items, refer to the "Setting Preferences" topic in the "Design Setup" chapter.

Rotation

Rotates the text string by 90 degrees.

Justification Area

Locates the text at the cursor, or X,Y coordinate location, in the Horizontal and Vertical planes as follows:

Horizontal	To the Right, at the Center, or to the Left.
Vertical	Up, at the Center, or Down.

Creating 2D Line Items

To add drafting objects such as polygons, circles, rectangles, paths, and arcs to polygons or paths, choose the Create 2D Line icon from the Drafting toolbox. This changes the default pop-up menu to a specific pop-up menu for creating 2D line items.

1. From the Drafting toolbox, choose the Create 2D Line icon.
2. Click the right mouse button to display a pop-up menu. Set one or more of the following values before you add the drafting object:

Shape	Choose to add a Polygon, Circle, Rectangle, or Path.
Width	Specify a width value to override the default.
Orthogonal	Adds segments in 90-degree increments.
Diagonal	Adds segments in 45-degree increments.
Any Angle	Adds segments at any angle.

Adding Polygons or Paths

To add polygons or paths, with or without arcs:

1. Choose the Create 2D Line icon from the Drafting toolbox.
2. Choose Polygon or Path from the pop-up menu.
3. Indicate the start point. A line follows the cursor movement.
4. Indicate the next corner.
5. Continue adding corners in the same manner. If you want to add an arc rather than a straight segment, choose Add Arc from the pop-up menu.
6. Press BackSpace or choose Del Corner from the pop-up menu to remove the last corner.
7. Double-click to finish adding. Polygons automatically close; paths terminate at the last-defined corner.

Adding Circles

To add circles:

1. Choose the Create 2D Line icon from the Drafting toolbox.
2. Choose Circle from the pop-up menu.
3. Indicate the center point of the circle. A circle outline follows the cursor movement.
4. Indicate the radius length to end the addition.

Adding Rectangles

To add rectangles:

1. Choose the Create 2D Line icon from the Drafting toolbox.
2. Choose Rectangle from the pop-up menu.
3. Indicate one corner of the rectangle. A box outline follows the cursor movement.
4. Indicate the diagonally opposite corner to end the addition.

Optionally, you can add multiple copies of a 2D line item after the initial item location is indicated. See the "Step and Repeat" topic in the "Design Operations" chapter for more information.

Modifying 2D Line Items

To change drafting objects, choose the Modify 2D Line icon from the Drafting toolbox. This changes the default pop-up menu to a specific pop-up menu for modifying 2D line items.

1. From the Drafting toolbox, choose the Modify 2D Line icon.
2. Select the line segment or corner to modify. The object is placed in Move mode.

   Use the pop-up menu to enable other 2D line modifications. If you choose a segment, the following modification functions are available:

Pull Arc	Converts a segment or corner into an arc.
Split	Divides a segment into two segments.
Del Segment	Deletes the line segment at the pick point.
Width	Changes the line width of the item.
Filled	Fills a closed polygon.
Solid Style and Dotted Style	Changes the line style to a solid or a dashed line. You cannot specify dotted lines for circles or arcs.
Orthogonal	Moves are made in 90-degree increments.
Diagonal	Moves are made in 45-degree increments.
Any Angle	Moves are made at any angle.

   If you choose a corner, the pop-up menu uses most of the options as selected segments, plus the following:

Del Corner	Deletes the corner. A line segment is created between the corner's original end points.

   3. Reposition the cursor and indicate a new location for the item.

Pulling Arcs

Pull Arc moves the selected line segment to create an arc, as the cursor is moved. Horizontal lines move in the vertical direction. Vertical lines move in the horizontal direction. Diagonal lines move in all directions, unless anchored by a line at the end of the diagonal. If an existing arc is selected, its end points remain fixed and the arc chord follows the cursor.

Use the Pull Arc command to convert a drafting segment or corner into an arc. The starting points of the drafting segment become the start and stop angle for the new arc.

1. Choose the Modify 2D Line icon from the Drafting toolbox.
2. Select the line for arc conversion.
3. Choose Pull Arc from the pop-up menu. An arc follows the cursor movement.
4. Indicate the arc to end the conversion.

Splitting Segments

Split converts a line segment into two separate segments, joined by a third segment perpendicular to the original segment.

All line segments in a 2D line must have the same line width and style. If you need to create a 2D element with more than one line width or style, create two separate 2D lines and use the Combine command to join them.

1. Choose the Modify 2D Line icon from the Drafting toolbox.
2. Select the line to split.
3. Choose Split from the pop-up menu. The new and divided segments follow the cursor movement.

   You can choose one of the following from the pop-up menu when splitting a segment:

Swap Corner	Selects the other corner or line segment created by the split.
Del Corner	Deletes the current corner.
Width	Changes the line width of the item.
Solid Style and Dotted Style	Changes the line style to a solid or a dashed line. You cannot specify dotted lines for circles or arcs.
Orthogonal	Moves are made in 90-degree increments.
Diagonal	Moves are made in 45-degree increments.
Any Angle	Moves are made at any angle.

4. Indicate the end point of the new segment to complete the split.

Combining 2D Line Items

To merge two or more 2D line or text items into a single complex 2D line item, choose the Combine icon from the Drafting toolbox. This changes the default pop-up menu to a specific pop-up menu for combining 2D line items.

Once combined, you can manipulate (move, rotate, copy, etc.) the group as a single item. This is useful for creating title blocks or other similar features.

1. Choose the Combine icon from the Drafting toolbox.
2. Choose Combine from the pop-up menu and select a line item.
3. Continue to select the items to combine. Selected line items change to the highlight color and selected text items disappear.
4. Choose Complete from the pop-up menu.

When you select any item in a complex group for move, copy, etc., all items are selected for the operation.

To delete or uncombine items from the group, choose Uncombine from the pop-up menu and select the line or text item to remove. After you select the item(s) from the complex group, choose Complete.

When combining 2D lines and free text, always select the 2D line first, then select and combine the text with the selected 2D line. If you try to select a text item before first selecting a 2D line, the message *No drawing item to add text to* appears.

Adding Drafting Items from a Library

To load a 2D line item from a library to the current schematic, choose the Add From Library icon from the Drafting toolbox. The Get Drafting Item from Library dialog box appears where you can choose a drafting item from the available libraries.

1. Choose the Add From Library icon from the Drafting toolbox.
2. Choose an item from the drop-down list box in the Get Drafting Item from Library dialog box. If the list does not contain the item you need, use the filter to search the available libraries.

   Use the wildcard convention, with or without leading characters, to expand or narrow the filter. Choose Apply to activate the filter.
3. Choose OK. The item follows the cursor movement.
4. Optionally, you can rotate or mirror the item before you indicate its location. Choose the Rotate, Mirror X, or Mirror Y command from the pop-up menu as often as needed to set the correct orientation of the item.
5. Indicate the item location.

PowerLogic searches only active libraries with the Lines option set to check the 2D element.

Adding Drafting Items to a Library

To save a *2D line item* or a complex 2D line item in the schematic to a lines library, choose the Save to Library icon from the Drafting toolbox.

1. Choose the Save to Library icon from the Drafting toolbox.
2. Select the item.
3. Type the item name and specify the library location in the Save to Library dialog box.
4. Choose OK.

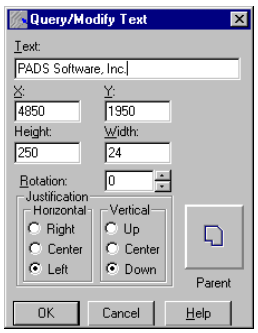
Query/Modify

Choose the Query/Modify icon to place PowerLogic in Query/Modify mode. Use this mode to get information on and modify objects in PowerLogic. When you select an object in Query/Modify mode, a dialog box or information window appears.

Query/Modify Text

Use the Query/Modify Text dialog box to modify free text or change its size, orientation or justification.

1. Choose the Query/Modify icon from the toolbar.
2. Select the text to modify.
3. Modify the text string and/or other options, then choose OK.



Text

Displays the selected text string. Modify the existing string or type a new text string.

X,Y Location

Displays the current X and Y location of the selected text string. Type new values to move the text string to a specified location.

Height, Width

Displays the current height and width of the selected text string. Type new values to change the height or width of the text string. Use Height/Width tab in the Preferences dialog box to change the default height or width of free text.

Rotation

Displays the current rotation value of the selected text string. The rotation can be either 0 or 90 degrees. Type a new value or use the arrows to change the degree of rotation.

Justification

Displays the current justification of the selected text string. Choose one of the option buttons to change the justification. Choose between horizontal alignments: Right, Center, or Left. Choose between vertical alignments: Up, Center, or Down.

Parent

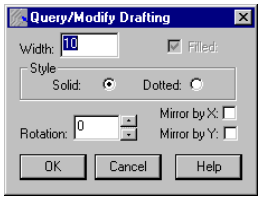
Displays the Query/Modify Drafting dialog box to modify the drafting object when you combine the selected text string with a drafting object.

Query/Modify Drafting Object

Use the Query/Modify Drafting dialog box to modify the line width, style, and orientation of selected drafting objects.

Note: Use the Drafting toolbox to modify the shape of 2D lines.

1. Choose the Query/Modify icon from the toolbar.
2. Select the drafting object to modify.
3. Modify the drafting object, then choose OK.



Width

The current line width of the selected drafting object. Type the new line width of the drafting object. Use Height/Width tab in the Preferences dialog box to change the default line width.

Filled

Creates a filled shape from a selected polygon. This option is grayed if you used Pull Arc to modify the polygon. This option is grayed and for all other drafting shapes.

Style

Displays the current line style of the selected drafting object. Choose an option button for a different line style. Choose from Solid or Dotted.

Rotation

Displays the current rotation value of the selected drafting object. Rotation can be 0 or 90 degrees. Type a new value or use the arrows to change the degree of rotation.

Mirror by X,Y

Mirrors the selected drafting object in the X (horizontal) or Y (vertical) direction. The point used when selecting the object is the also the point of rotation.

Chapter 9
OLE in PowerLogic

This chapter discusses PowerLogic's object linking and embedding capabilities. This chapter also discusses the Automation features available using Visual Basic scripting. The topics in this chapter include:

OLE Functions in PowerLogic
OLE Terminology
OLE, Object Linking and Embedding
Displaying OLE Objects
Selecting OLE Objects
Moving and Sizing OLE Objects
Cutting, Copying, and Pasting OLE Objects
Editing OLE Links
OLE Visual Editing
Converting OLE Objects and other OLE-specific Editing Commands
Saving OLE Objects
Deleting OLE Objects
OLE and Print/Plot
OLE PowerPCB Connection
Visual Basic Scripting

OLE Functions in PowerLogic

PowerLogic's object embedding capabilities allow design engineers to embed an object into a PowerLogic design file and hold it within its framework. You can also link objects to their source so linked objects automatically update each time you open the PowerLogic database.

OLE functions in PowerLogic include:

- Object Linking and Embedding
- Lets you link or embed objects, such as Microsoft® Word documents and Microsoft Excel spreadsheets, within a PowerLogic design.
- OLE Cross-probing with PowerPCB
- Lets you link to PowerPCB, cross-probe, perform netlist comparisons, and update design rules using OLE.

OLE Terminology

Embedded Objects

When an object is embedded, a copy of the object, including all of its data, is contained within the framework of and is a part of the PowerLogic schematic.

Linked Objects

When an object is linked, a copy of the object, including all of its data, is contained within the framework of and is a part of the PowerLogic schematic; like an embedded object. However, the object is also linked to its source (the original object that was copied). Whenever you open the PowerLogic schematic that contains the linked object, the object checks the source to see if it changed since you last saved the schematic. If the source changed, then the linked object automatically updates.

OLE Automation

OLE Automation is a way for heterogeneous applications to communicate with each other. You can also use OLE Automation to communicate with standard Microsoft Office applications such as Word and Excel.

Applications can only communicate if one is an OLE Automation Server. PowerLogic is not an OLE Automation Server. PowerPCB, however, is. Therefore, PowerLogic can successfully communicate with PowerPCB.

Visual Editing

Visual Editing occurs when the source application for a linked or embedded OLE object opens within PowerLogic. The source application's menu system appears within PowerLogic; this is In-place Visual Editing.

Sizing handles

Small, black squares that appear at the corners and along the sides of a rectangular area that surrounds a selected object.

OLE, Object Linking and Embedding

PowerLogic now lets you insert other files or other applications as linked or embedded objects within a schematic. You can insert Microsoft Word documents, insert a Microsoft Excel spreadsheet containing a Bill of Materials, and insert video or audio clips. PowerLogic does not need to understand the format of the inserted object; PowerLogic communicates with the application that created the file and that source application tells PowerLogic what information to display and how to display it.

Note: You cannot insert or modify OLE linked or embedded objects in the Part Editor.

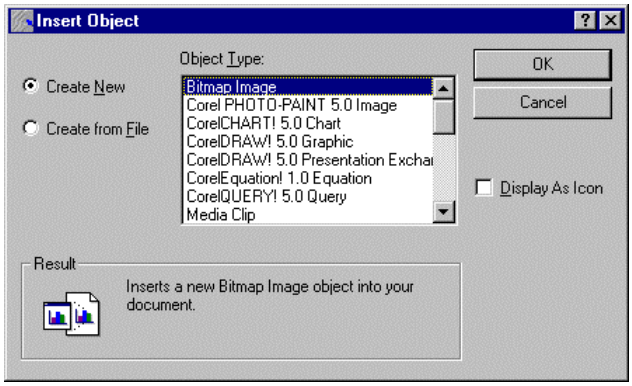
To insert a linked or embedded object in PowerLogic:

1. Choose Edit/Insert New Object. The Insert Object dialog box appears.
2. Choose whether to Create New or Create from File. Create New inserts a new OLE object. Create from File inserts an existing file as an OLE object.
3. If you chose Create New, choose the type of OLE object you want to create. If you chose Create from File, choose the file you want to insert as an OLE object.
4. If you chose Create from File, choose whether to link the inserted object to the original file. If you choose not to link the object, it is an embedded object.
5. Choose whether to display the linked or embedded object as an icon.
6. Choose OK to insert the linked or embedded object in PowerLogic.

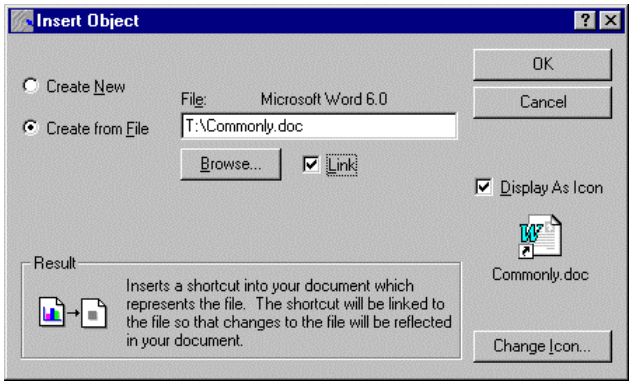
If you insert a new object whose source application is an OLE Automation Server, that application opens inside of PowerLogic, but runs in the background. The source application's toolbar takes over PowerLogic's toolbar. You can then work with the source application as you would if it was started outside of PowerLogic. This is referred to as Visual Editing.

Click outside of the object and the PowerLogic toolbar takes over again. You can continue to design in PowerLogic. The source application continues to run in the background, therefore, you can click on the object and work in the source application at any time.

Insert Object Dialog Box: Create New



Insert Object Dialog Box: Create from File



Create New

Creates a new linked or embedded object using the appropriate application. The application starts within PowerLogic. For example, if you want to insert a Word document, Word appears to open within PowerLogic. Word menus and toolbars replace the PowerLogic menus and toolbars inside the PowerLogic window.

When you choose Create New, you must also choose the type of object to create.

Note: You can only create new OLE objects using applications that are installed and registered on your system and are OLE servers. Most OLE servers are Microsoft products, such as Word or Excel.

Create from File

Creates an OLE linked or embedded object based on an existing file. When you choose Create from File, choose the file from which to create the OLE object, and choose whether to link the OLE object to the file.

Object Type

Choose the type of OLE object you want to insert. The Object Type choices are based on which products are currently installed on your system.

File

The file to insert. Only a copy of the file is inserted; the original remains unchanged. Choose whether to Link the OLE object to the file; linked OLE objects are automatically updated when you modify the original file. If you do not choose to link the OLE object, it is embedded and does not automatically update.

Browse

Use the Browse button to search for a different file.

Link

Links the inserted OLE object to the original file. When you modify the original file, linked OLE objects automatically update. If you do not choose to link files, OLE objects are embedded and do not reflect any updates you make to the original file.

Display As Icon

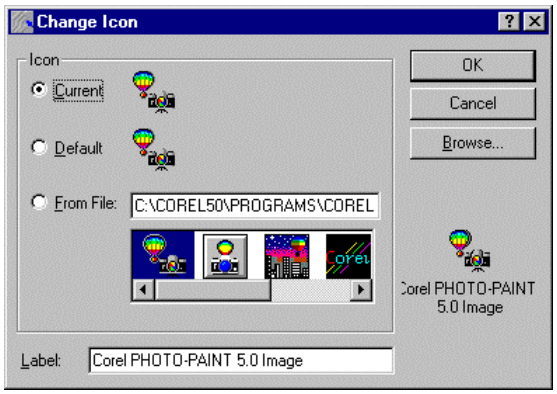
Displays the inserted OLE object as an icon. Leave this option off to display an entire object. For example, to display an actual Word document instead of a Word icon. If the object's source application is not registered on your system, then objects can only be displayed as icons.

Result

Displays the results of the selected options.

Change Icon

Changes the icon that represents the OLE object using the Change Icon dialog box.



Current

Uses the current icon assignment for the application as the OLE object icon.

Default

Uses the default icon assignment for the application as the OLE object icon.

From File

Displays the name of the file that contains the icons that appear in the Icon List below. Use Browse to choose a file that contains the icons you want to use to represent the OLE linked or embedded object. Once you locate the file, choose the icon to use from the list below. The icon you choose appears to the right of the From File area.

Icon List

Choose the icon you want to use to represent the OLE linked or embedded object. To use different icons, use From File and Browse to locate another file containing the icons you want to use.

Label

Type a description of the icon you want to use. This description appears under the linked or embedded object icon.

Displaying OLE Objects

PowerLogic does not understand the format of an inserted object; it only communicates with the application that created the OLE linked or embedded object to display the information. If the application that created the linked or embedded OLE object is installed and registered on your system, then PowerLogic calls upon that application to display the OLE object in PowerLogic as it would appear in the source application. For example, a Word document can appear within PowerLogic and the Word toolbars appear within PowerLogic.

If the source application is not installed and registered on your system, then PowerLogic can only display the inserted OLE object as an icon. It cannot open or display the OLE object as it would appear in the source application. PowerLogic also displays the OLE object as an icon if the object is an application.

OLE and View Menu Commands

You can use all of the View menu commands with OLE linked or embedded objects; you can zoom into and zoom out of the objects.

Turning the Display of OLE Objects Off

You can turn the display of OLE objects off using the Display OLE Objects check box on the Global tab of the Setup/Preferences dialog box. You may want to turn off the display of OLE objects when PowerLogic contains many linked or embedded objects because the redraw speed can decrease. Redraw speed noticeably decreases if the OLE Linking and Embedding servers, or source applications, which actually display the OLE items are not optimized for remote display.

Redrawing a Screen Containing OLE Objects

When PowerLogic redraws, PowerLogic components are redrawn first, then OLE linked or embedded objects are redrawn. OLE objects always redraw in the same order and always redraw after PowerLogic objects; therefore, they always appear on top of PowerLogic components.

You can also choose to update linked and embedded objects when you redraw the work area using the Redraw OLE Objects check box on the Global tab of the Setup/Preferences dialog box. This option is grayed when no OLE objects exist.

Changing the OLE Object Background Color

OLE linked or embedded objects are displayed with a solid white background. You may, in some cases, prefer to display the OLE object with a transparent background; for example, since a bitmap already contains a background, you may prefer to use a transparent background. If your object is a Word document, then you may prefer a white background because black text on a transparent background results in black on black, or an invisible object.

To change the background color of an OLE object:

1. Select the OLE object.
2. Choose White Background from the pop-up menu to change the background color. A check next to the command indicates that the object will use a white background.

Selecting OLE Objects

Select and manage OLE linked or embedded objects just like you manage non-text Word items. Click on the OLE object to select it.

Some limitations exist for selecting OLE objects:

- You cannot select more than one OLE object at a time.
- You cannot use area select to select OLE objects.
- Commands apply to selected OLE objects only, even if you also select PowerLogic objects. OLE objects have selection priority over PowerLogic components.

Note: To select PowerLogic items under an OLE object, move the OLE object.

When you select an OLE object:

- A pop-up menu is available that lists all commands that you can apply to the OLE object. Click the right mouse button when the object is selected to open the pop-up menu.
- The object is selected just as a non-text object in Word is selected, using a rectangular area with sizing handles to indicate that it is selected. OLE objects do not use the PowerLogic method of highlighting to indicate selection state.

Moving and Sizing OLE Objects

Move and resize OLE linked or embedded objects just as you resize non-text objects in Word.

To move an OLE object:

1. Select the object.
2. Click and hold the left mouse button.
3. Move the cursor to move the object.
4. Release the mouse button once the object is in the correct location.

To size an OLE object:

1. Select the object.
2. Click and hold the left mouse button on one of the sizing handles.
3. Move the cursor; the object changes size according to the cursor movements.
4. Release the mouse button when the object is sized correctly.

Cutting, Copying, and Pasting OLE Objects

Editing an OLE linked or embedded object is similar to editing PowerLogic objects. You can cut, copy, and paste OLE objects using the Cut, Copy, and Paste commands from the Edit menu. You must select the OLE object before you can edit the object. You can copy objects from one sheet to another.

Note: You cannot cut, copy, or paste OLE objects when in the Part Editor.

To copy, cut, or paste an OLE object:

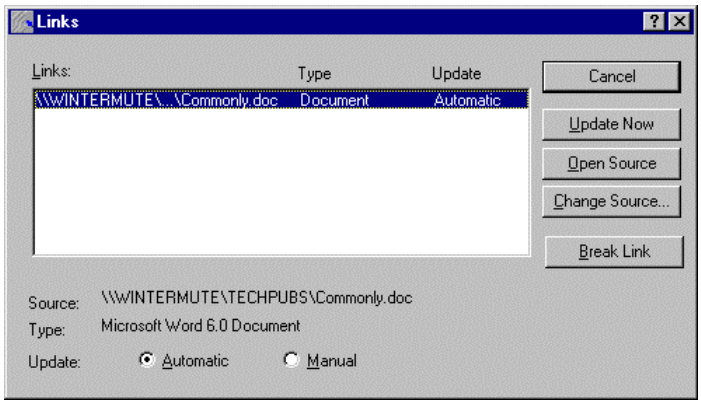
1. Select the OLE object to cut, copy, or paste.
2. Choose Cut, Copy, or Paste from the Edit menu.

Editing OLE Links

You can edit the link of a linked OLE object. Editing the link allows you to update the link, open the original object source, change the original object source, or break the link with the object source making an embedded OLE object. You can also choose to update the object automatically or using a manual command.

To edit the link of an OLE object:

1. Choose Edit/Links. The Links dialog box appears.
2. Select the link you want to edit from the list.
3. Choose the options you want to use or modify.
4. Choose Close to close the Links dialog box. You cannot cancel changes you make in the Links dialog box.



Links

Lists the current links for all linked objects in the current design. The list also includes the source path and file for the object.

Type

Lists what kind of file the source file is, for example, a document. The object is linked to this file type.

Update

Lists the Update setting assigned to the object. Choose between assigning an Automatic or Manual update to the linked file. Change the setting using Update.

Update Now

Updates any linked objects that you select in the Links list.

Open Source

Opens the source file in the source application.

Change Source

Opens the Change Source dialog box where you can choose a different source for the object selected in the Links list.

Break Link

Breaks the object's link to the source file. The object is embedded. After you break an object's link, you cannot reconnect the link.

Automatic Update

Updates the object's data when you open the PowerLogic file that contains the object and when the source file changes.

Manual Update

Updates the objects selected in the Links list when you choose Update Now from this dialog box.

Source File

Lists the path and filename of the source file.

OLE Visual Editing

Visual Editing occurs when the source application for a linked or embedded OLE object opens within PowerLogic. The source application's menu system appears within PowerLogic; this is In-place Visual Editing. You can also visually edit an OLE object by opening the source application and editing the object in the environment in which it was created. You can visually edit OLE objects in two ways:

- Double-click the OLE object to edit directly within PowerLogic. The source application's menu system appears within PowerLogic.
- Ctrl-double-click the OLE object to edit in the source application. The source application opens and you edit the object in the original environment.

Note: Once the object opens within either PowerLogic or the source application, you can edit its content as you normally would using all of the source application's commands and tools.

Converting OLE Objects and other OLE-specific Editing Commands

Once you insert an OLE object, the name of the object appears at the bottom of the Edit menu. For example, if you insert a video clip, Video Clip Object appears at the bottom of the Edit menu. If you highlight the Video Clip Object command, another menu appears listing all commands you can perform on the linked or embedded OLE object. For a Video Clip Object, you can choose to Play, Edit, or Open the object.

Note: The commands that appear for each object depend on the object type; therefore, a Word object will not have the same options that a video clip has.

OLE objects have some common commands. One of those commands is Convert. You can convert an OLE object to another object. You can also convert the OLE object from displaying as an icon to displaying the actual object; for example, the Word document instead of an icon.

Note: The object's source application determines to what the object can be converted.

Edit

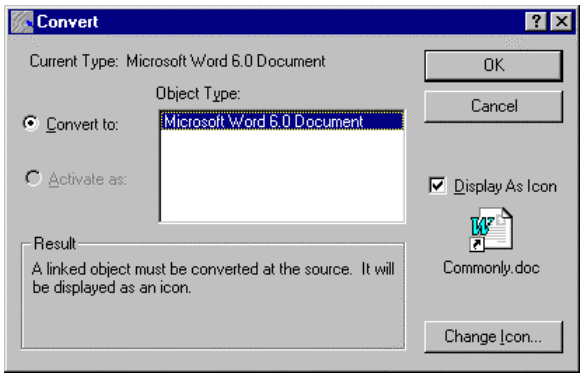
Choose Edit to edit the linked or embedded OLE object in PowerLogic. You can edit the object using all of the source application's commands and tools. When you are finished editing, choose File/Close. You return to PowerLogic and the OLE object is updated.

Open

Choose Open to open the linked or embedded OLE object in the source application. You can then edit the object within the source application. When you finish editing, choose File/Exit. You return to PowerLogic and the OLE object is updated.

Convert

Choose Convert to convert a linked or embedded OLE object to another type of object; for example, convert a Word document to a Word picture. The Convert dialog box appears.



Current Type

Lists what the current type of the inserted object is.

Convert To

Converts the selected OLE linked or embedded object to the application format selected in the Object Type box. Use Convert To to change the file format of the object so that you can open it in the applications you use.

Activate As

Opens the selected OLE linked or embedded object as the Object Type selection. After you activate and edit the object, it returns to the current type.

Object Type

The type of object to which you can convert the currently selected linked or embedded OLE object. The currently selected OLE object can then open for editing in the source application.

Saving OLE Objects

Linked and embedded objects are automatically saved when you save a PowerLogic design. OLE objects are saved in a separate file with an .ole extension. By default, the .ole file is given the same name as the .sch file it is associated with, however _sch is added at the end of the filename to indicate that the .ole file goes with a schematic instead of a PCB design. _pcb is added to the name of the schematic containing OLE objects. This allows you to easily manage project directories where the schematic and design files have the same name and contain OLE objects. For example, if you insert OLE objects into and save a schematic named design1.sch, the design1_sch.ole file is also created. If you insert objects into and save a PCB design named design1.pcb, the design1_pcb.ole file is also created. Both files (design and OLE) are located in the same directory. When you open the .sch file, the related .ole file is also automatically opened.

Note: Remember both the .sch file and the .ole file when you copy, move, or attach them in any way.

PowerLogic .ole files can be opened in other applications that understand the .ole file format. For example, if you insert a Word document into PowerLogic and then save the Word object, you can later open Word and open the Word documents stored in the PowerLogic .ole file.

To save a new PowerLogic design in a .sch file and save OLE information in an .ole file:

1. Choose File/Save As. The Save As dialog box appears.
2. Type a name for the design file you are saving.
3. Choose the location for the file.
4. Choose Save. The .sch and .ole files are created.

Note: You can also import the .ole file into another PowerLogic file. For example, if you save design1.sch and it has OLE objects in it, design1.sch and design1.ole are saved. You can then import design1.ole into design2.sch if you want to share the OLE objects between multiple files. See the "OLE Import/Export" topic in the "File Operations" chapter for more information.

Deleting OLE Objects

You can delete OLE linked or embedded objects. To delete an OLE object:

1. Select the OLE object to delete.
2. Choose Edit/Delete or press the Delete key. You are asked to confirm the deletion.
3. Choose Yes to delete the object; choose No to return to PowerLogic without deleting the object.

Note: When you delete a sheet with OLE objects on it, all of the OLE objects are also deleted.

OLE and Print/Plot

You can print OLE linked or embedded objects to any Windows-supported printer or plotter. You cannot photoplot or pen plot OLE objects. Also, OLE objects do not appear when previewing prints. See the Plot Options dialog box for more information.

OLE PowerPCB Connection

You can link to and cross-probe PowerPCB using OLE. This new function allows you to receive and send selections, start a new PowerPCB session or document, and manage certain net preferences. This feature also automates certain tasks for you, such as netlist comparisons and rules exports. Once you connect to PowerPCB, you can select objects in PowerPCB and the object is automatically highlighted and centered into the PowerLogic work area.

To initiate cross-probing:

1. Choose Tools/OLE PowerPCB Connection.

   If PowerPCB is already running, PowerLogic automatically connects to it. Proceed to Step 3. If PowerPCB is not running, the *Connect with New PCB Session or Open Existing PCB File* message appears.

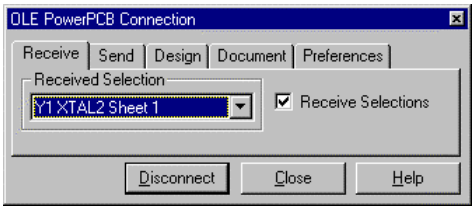
2. Choose New to start a new session without loading a file or choose Open to open an existing file in a new PowerPCB session. If you choose Open, you need to locate the file to open.
3. The OLE PowerPCB Connection dialog box appears. You are connected to PowerPCB. Choose the tab that represents the area you want to set. Choose between:

Receive tab	Manages received selections.
Send tab	Manages selections to send.
Design tab	Automates the ECO, File Compare, and other functions using the OLE communications and PowerPCB OLE Automation tools.
Document tab	Allows you to open a file or create a new file in PowerPCB. This tab also displays the name of the current PowerPCB file to which PowerLogic is connected.
Preferences tab	Manages nets.

4. When you choose the tab, the options for that tab appear. Set the options you want to change.
5. Proceed with cross-probing.

OLE Receive Tab

The Receive tab manages received selections from PowerPCB.



OLE Received Selection

Contains a list of items selected in PowerPCB if the Receive Selections check box is on. Items are listed in the order in which they are received. When you choose an item in this list box, it appears in the center of the PowerLogic work area.

OLE Receive Selections

Enables PowerLogic to automatically locate the item corresponding to the PowerPCB selection. When this check box is off, items are not listed in the Received Selection list box.

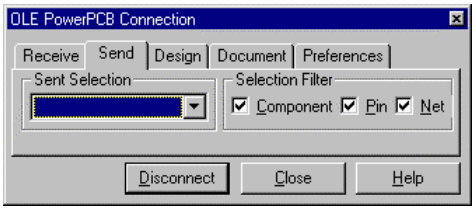
OLE Connect/Disconnect

This button changes depending on whether you are currently connected to PowerPCB.

Connect	Behaves exactly like choosing Tools/OLE PowerPCB Connection. See the "OLE PowerPCB Connection" topic for more information.
Disconnect	Disconnects from PowerPCB. The *Disconnect from PowerPCB?* message appears before disconnecting. Choose OK to disconnect.

OLE Send Tab

The Send tab manages the selections to send to PowerPCB.



OLE Sent Selection

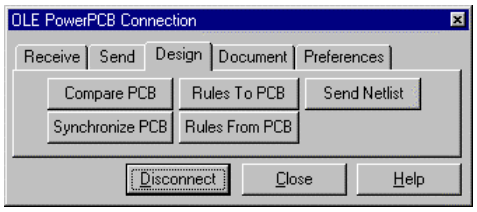
Contains a list of items selected in PowerLogic. Items are listed in the order in which they are selected.

OLE Selection Filter

Contains the Component, Pin, and Net check boxes. Turn a check box on to allow selection of these items in PowerLogic and send selections of these items to PowerPCB.

OLE Design Tab

The OLE Design tab provides automatic methods of performing many cross-probing actions.



Compare PCB

Compares the netlist in the current PowerLogic schematic to the netlist in the current PowerPCB design. This comparison uses the Check ASCII tool. If differences exist, they are reported in PowerPCB's default text editor. See the "Check ASCII" topic in the PowerPCB On-line Help for more information.

Synchronize PCB

Compares the netlist in the current PowerLogic schematic to the part and netlist in the current PowerPCB design. Then the PowerPCB design is updated. The *Design synchronization complete* message appears when the design is updated. This comparison uses the ECOGEN tool. See the "ECOGEN" topic in the PowerPCB On-line Help for more information.

Rules to PCB

Exports PowerLogic design rules to PowerPCB. PowerPCB rules are updated.

Rules from PCB

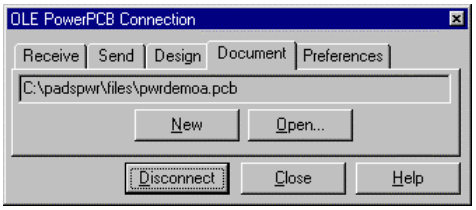
Exports PowerPCB design rules to PowerLogic. PowerLogic design rules are updated.

Send Netlist

Exports a netlist with rules from the current PowerLogic schematic to the current PowerPCB design. PowerPCB is updated.

OLE Document Tab

The Document tab allows you to open a file or create a new file within PowerPCB. The name of the PowerPCB file to which PowerLogic is connected also appears in this tab.



New PCB File

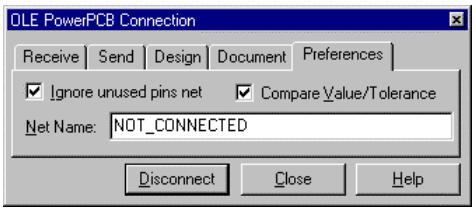
Creates a new file within PowerPCB. PowerLogic automatically connects to this file.

Open PCB File

Allows you to choose an existing PowerPCB file to open. You need to locate the file to open. PowerLogic automatically connects to this file.

OLE Preferences Tab

The Preferences tab manages the unused pins net created by SPECCTRA and netlist comparison preferences.



Ignore Unused Pins Net

Turn this option on to ignore the unused pins net in the original design, essentially preserving the existing unused pins net in the design you're updating. Otherwise, the unused pins net is deleted. Specify the net name when you turn this option on.

To compare a schematic to a design, first export the schematic to ASCII format.

Net Name

Specify the name of the unused pins net in SPECCTRA. The default name is NOT_CONNECTED. See the "Unused Pins Net" topic in the SPECCTRA Translator On-line Help for more information.

For Output of value tolerance, see this option in the Net List topic in Reports.doc. This dialog box, which appears when you choose Netlist to PCB, contains the same option, the same meaning.

Visual Basic Scripting

PowerLogic provides an internal Visual Basic® scripting capability, or Visual Basic Engine, which makes the Automation features within PowerLogic more accessible to everyone; developer and non-developer.

Visual Basic is a simple scripting language developed by the Microsoft Corporation in the late 1980s to provide users with a unified language in Windows 95 and Windows NT. More and more Windows applications like PowerLogic include Visual Basic capabilities, such as Word and Excel, to allow users to customize these applications using a standard scripting language.

The Visual Basic Engine includes the entire development environment required to develop Visual Basic scripts including an editor, a debugger, an interpreter, a tracer, variable watch, a dialog editor, new/open/save/print capabilities, and an Automation object browser. These capabilities ensure that every user can jump into Visual Basic scripting without any previous knowledge of programming. To open the Visual Basic editor, choose Tools/Visual Basic Scripting/Visual Basic Editor.

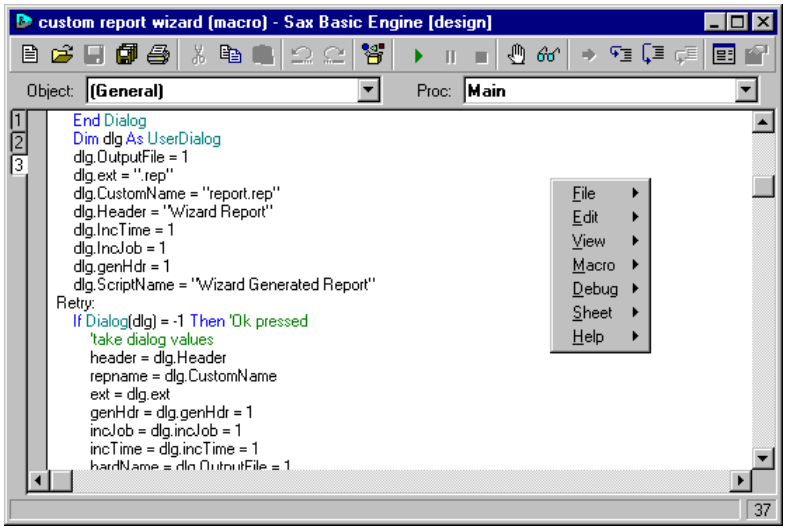
PADS' Visual Basic editor is the Sax Basic Engine™, by Sax Software Corporation. In addition to the Visual Basic editor, PowerLogic has a script manager that lets you manage your existing Visual Basic scripts. Using the script manager, you can load your most frequently used scripts and run them independently from the editor itself. To open the Visual Basic Scripts dialog box, choose Tools/Visual Basic Scripting/Visual Basic Scripts.

Note: While a Visual Basic script is running, a third command, Stop *scriptname*, appears on the Tools/Visual Basic Scripting menu along with Visual Basic Editor and Visual Basic Scripts. Choose Stop *scriptname* to halt the execution of the currently running script.

Sax Basic Engine Dialog Box

The Sax Basic Engine dialog box provides access to the Sax Basic Engine Visual Basic editor. You can design and develop scripts that add to, replace, enhance, or customize existing PowerLogic features. To open the Visual Basic editor, choose Tools/Visual Basic Scripting/Visual Basic Editor.

Scripts written in the Visual Basic editor comply with all Microsoft requirements in terms of Visual Basic syntax; therefore, you can play these scripts in any other Visual Basic interpreter, such as Word or Excel. However, you cannot run all Visual Basic scripts created outside of the Sax Basic Engine within the Sax Basic Engine because the Sax Basic Engine is a subset of Visual Basic. For example, you cannot run the Automation samples within the Sax Basic Engine.



For more information on the Visual Basic editor and language, see the Sax Basic Editor On-line Help.

Editor Colors

This Visual Basic editor displays source code using different colors. The color is context-sensitive; when you place the cursor on the text and press F1, the correct help file opens to the correct help topic. For example, if the cursor is on a PowerLogic Automation Object (purple) when you press F1, the Automation Server On-line Help appears. The text colors represent:

Blue	Visual Basic Keywords
Black	User Variables
Cyan	Visual Basic Functions
Purple	PowerLogic Automation Objects or Members
Red	Errors
Green	Comments

Editor Menus

To open the editor pop-up menu, right-click anywhere on the Sax Basic Engine dialog box. For more information on the Visual Basic editor dialog box features, see the Sax Basic Editor On-line Help.

Editor Toolbar

New

Creates a new Visual Basic script. There are four types of scripts: scripts, code modules, object modules, and class modules. See the Sax Basic Editor On-line Help for more information.

You can create a script that calls another script. For example, ScriptA can call ScriptB.

You can also create a script that runs a series of scripts, or a "master" script. For example:

```
'$Include:  "scriptA.bas"
'$Include:  "scriptB.bas"
'$Include:  "scriptC.bas"
Sub Main
   Call scriptA
   Call script B
   Call script C
End Sub
```

Open

Loads a new script file into the editor. You can load up to nine scripts at a time.

Save

Saves a script file to the \padspwr\logicole\vbscripts directory with a .bas extension.

Save All

Saves all scripts currently loaded.

Print

Prints a script file.

Cut

Cuts the highlighted code to the Clipboard.

Copy

Copies the highlighted code to the Clipboard.

Paste

Pastes the selection from the Clipboard into the text window.

Undo

Reverses the last action taken.

Redo

Reverses an undo action.

Object Browser

Opens the Object Browser dialog box. This dialog box shows all objects available in PowerLogic (such as Component, Application, and Pin) and lists the properties and methods for each object.

Run

Runs a script.

Pause

Pauses a running script.

Stop

Ends a running script.

Breakpoint

Places or removes a break point.

Show Current Statement

Moves the cursor before the Visual Basic statement that will be executed next. Used during debugging when execution is paused.

Watch

Watch a variable.

Step Into

Enters the line of code at the cursor location.

Step Over

Steps over the line of code at the cursor location.

Step Out

Steps out of the current function.

Dialog Editor

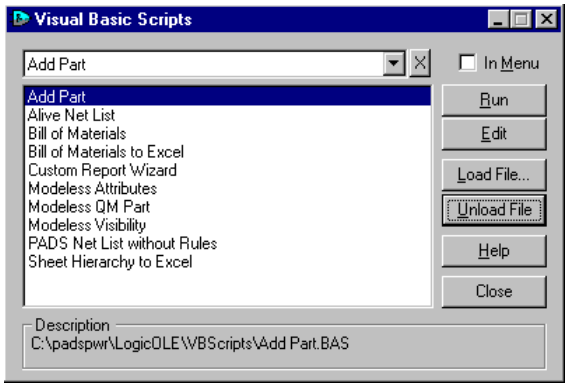
Opens the dialog box editor.

Options

Available only for class modules.

Visual Basic Scripts Dialog Box

The Visual Basic Scripts dialog box provides easy access to your Visual Basic scripts. In this dialog box, you can load and run your most commonly used scripts. You can also unload those scripts you don't use often. This dialog box only runs existing scripts. To open this dialog box, choose Tools/Visual Basic Scripting/Visual Basic Scripts.



Run

Runs the currently selected Visual Basic script. Select a script from the drop-down list box or from the list box and choose Run. Scripts are compiled when you run them.

You can not run multiple scripts at the same time. During script execution, Run is grayed.

If the selected script has a compiling error, it automatically opens in the Visual Basic editor for correction.

Edit

Opens the Sax Basic Engine dialog box where you can edit the selected Visual Basic script.

In Menu

Adds visual basic scripts to the Tools/Visual Basic Scripting menu, allowing you to run often-used scripts directly from the menu without having to open the Visual Basic Scripts dialog box.

Load File

Loads Visual Basic scripts into the list box. You can load up to 32,767 scripts. Scripts are not compiled when they are loaded: they are compiled when you run them.

This list of scripts you load into this dialog box is saved in the VBScripts.ini file, so these scripts load every time you open the Visual Basic Scripts dialog box.

Unload File

Unloads the selected Visual Basic script from the dialog box.

Reduce/Expand

You can use this dialog box in a reduced mode. When you click on the X button next to the Script drop-down list box, the lower half of the dialog box "disappears"; leaving only the top left corner of the dialog box for interaction. Choose the X button again to expand the dialog box. When in reduced mode, choose a script from the drop-down list box and press Enter, or double-click on a script in the drop-down list box.

Script List

Lists the loaded Visual Basic scripts. Both the Script list box and the Script drop-down list box are synchronized.

Script Drop-Down List

Lists the loaded Visual Basic scripts. Both the Script list box and the Script drop-down list box are synchronized.

The drop-down list box is most useful when you work with this dialog box reduced.

Visual Basic Sample Scripts

The following three lists provide descriptions of the Visual Basic scripts provided by PADS.

In the first list, samples 00 through 09 provide an overview of Visual Basic for users who don't have experience with Visual Basic scripts. Samples 10-11 provide small PowerLogic features that add, enhance, or customize a PowerLogic feature. These files are located in the \padspwr\logicole\vbscripts folder.

The second list describes Visual Basic scripts, which are equivalent to existing RGL reports. These files are located in the \padspwr\logicole\vbscripts\rgl samples folder.

The third list describes advanced Visual Basic scripts. These files are located in the \padspwr\logicole\vbscripts folder.

All sample scripts are commented.

Visual Basic Sample Scripts

00 What is a VB Script.BAS	Empty script demonstrating what a Visual Basic script is and how to define it.
01 Using a Message Box.BAS	Demonstrates how to display an OK dialog box.
02 Using a Variable.BAS	Demonstrates a Visual Basic variable: how to assign a value and how to get its value.
03 Using a VB Function.BAS	Demonstrates how to use a standard Visual Basic function and display its result in a message box.
04 Using a powerLogic Function.BAS	Demonstrates Visual Basic interaction with a PowerLogic Automation function.
05 Using If and Then Statements.BAS	Demonstrates the If, Then statements.
06 Using a Custom Dialog1.BAS	Demonstrates a simple dialog box using the Visual Basic dialog editor.
07 Using a Custom Dialog2.BAS	Demonstrates a standard dialog box using the Visual Basic dialog box editor.
08 Using a Custom Dialog3.BAS	Demonstrates a complex dialog box using the Visual Basic dialog box editor.
09 Using it All Together.BAS	Provides a "real life" example. Lists all design files in the \padspwr\files directory. Selecting a file from the list will open that file in PowerLogic.
10 List Of Comps and Nets.BAS	Lists all components and nets.
11 Select by Pin Count.BAS	Allows you to enter a number of pins. All parts with that number of pins are selected.

Visual Basic Sample Scripts/RGL Reports

The following list describes Visual Basic scripts, which are equivalent to existing RGL reports. These files are located in the \padspwr\logicole\vbscripts\rgl samples folder.

RGL.BAS	Contains a library of functions, which is used by the other scripts in this group; the scripts in this group must contain RGL.BAS to function.
Net Statistics.BAS	Lists all nets in the schematic and identifies any questionable nets.
Part Statistics.BAS	Lists all parts in the schematic.
Unused Gates.BAS	Lists all unused gates in the schematic.
Unused Pins.BAS	Lists all unused pins in the schematic.
Unused.BAS	Lists all unused pins and gates in the schematic.

Visual Basic Sample Scripts/Advanced

The following list describes advanced Visual Basic scripts. These files are located in the \padspwr\logicole\vbscripts folder.

Add Part.BAS	Adds a new gate to the schematic.
Alive Net List.BAS	Creates a netlist in Excel, which allows you to cross-probe between PowerLogic objects and the Excel cells containing the object names.
Bill of Materials.BAS	Produces a bill of materials in a user-customizable format.
Custom Report Wizard.BAS	Generates a Wizard dialog box, which you use to create a Visual Basic report.
Modeless Attributes.BAS	Generates a modeless dialog box, which you use to manage part attributes.
Modeless Qm Part.BAS	Generates a modeless dialog box, which you use to manage part and gate properties.
Modeless Visibility.BAS	Generates a modeless dialog box, which you use to manage gate and attribute visibility.
PADS Net List Without Rules.BAS	Generates a netlist report in ASCII format.
Sheet Hierarchy to Excel.BAS	Creates a sheet hierarchy report in Excel, which allows you to cross-probe between PowerLogic objects and the Excel cells containing the object names.

Chapter 10
Design Rules and Layers

This chapter discusses how to set up design rules and layers. Once you set up your design rules, you can export them to your layout tool, or you can import rules from your layout tool. The topics in this chapter include:

Setting Design Rules
Export Rules to PCB
Import Rules from PCB
Setting Layers

Setting Design Rules

Design Rules lets you assign general routing constraints for your design, such as trace width and spacing. An option in the netlist report lets you pass established design rules to PowerPCB along with the connectivity and parts information.

Design rules are separated into three categories:

Clearance rules	Sets the minimum allowable airgap between various object types in the design, such as trace to trace and via to trace.
Routing rules	Assign or prohibit via types, specify length minimization types, and allow or prohibit routing.
High-speed rules	Sets the minimum and maximum parameters for advanced design rules such as parallelism, delay, capacitance, and others.

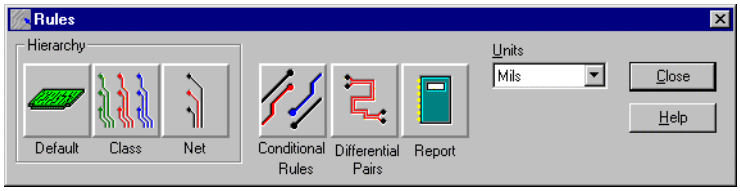
Information for each category is entered and edited in separate dialog boxes that you can access through the Rules dialog box. To set rules:

1. Choose Setup/Design Rules. The Rules dialog box appears.

   Use the Rules dialog box to enter item-to-item Clearance rules, routing guidelines, and values for the optional High-Speed checking commands.

2. Choose which rules hierarchy level you want to set or modify rules for.

When you specify one of the hierarchical levels described below, you can access the Clearance, Routing, or High-Speed dialog boxes.



Default

Opens the Default Rules dialog box where you set Default rules. See the "Default Rules" topic in this chapter for more information.

Class

Opens the Class Rules dialog box where you set Class rules. See the "Class Rules" topic in this chapter for more information.

Net

Opens the Net Rules dialog box where you set Net rules. See the "Net Rules" topic in this chapter for more information.

Conditional Rules

Opens the Conditional Rules dialog box where you set Conditional rules. See the "Conditional Rules" topic in this chapter for more information.

Differential Pairs

Opens the Differential Pairs dialog box where you set rules for Differential Pairs. See the "Differential Pairs" topic in this chapter for more information.

Report

Opens the Rules Report dialog box where you can create reports of your different rule settings. See the "Rules Report" topic in this chapter for more information.

Units

Indicates the unit of measure for passing rules to PowerPCB: mils, metric, or inches.

Rules Hierarchy

In the rules hierarchy, certain rules have precedence over other rules in the hierarchical rules order. For example, a net rule overrides a class rule and a class rule overrides a default rule.

Default	Rules which apply to an object if there are no other individually defined rules.
Class	Rules for a collection of nets, called a *class,* which need identical rules.
Net	Rules for a specific net.

Choose the appropriate icon in the Rules dialog box to define rules for a particular level in the hierarchy.

Hierarchical Rules Order

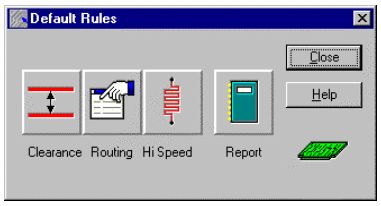
This section lists the order of precedence for all rules from least to most specific. Default, 1, represents the lowest level of the hierarchy with the least amount of precedence. At the opposite end of the order is Net against Net with Levels, which is the highest level of the hierarchy and has the highest possible precedence. It represents the most specific rule you can assign to an object in PowerLogic.

1 Default
2 Default with Level
3 Class
4 Class with Level
5 Net
6 Net with Level
7 Class against Class
8 Class against Class with Level
9 Net against Class
10 Net against Class with Level
11 Net against Net
12 Net against Net with Level

Default Rules

Use the Default Rules dialog box to define rules which apply to all objects that are not included in any other rule within the hierarchy.

1. Choose Setup/Design Rules. The Rules dialog box appears.
2. Choose the Default icon. The Default Rules dialog box appears.



Clearance

Opens the Clearance Rules dialog box where you set default Clearance rules. See the "Clearance Rules" topic in this chapter for more information.

Routing

Opens the Routing Rules dialog box where you set default Routing rules. See the "Routing Rules" topic in this chapter for more information.

High-speed

Opens the High-Speed Rules dialog box where you set default High-Speed rules. See the "High-speed Rules" topic in this chapter for more information.

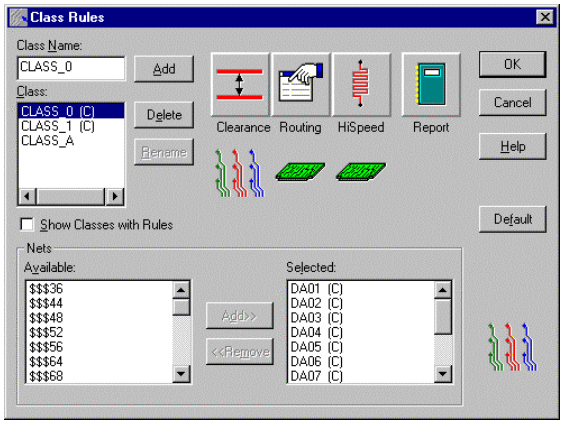
Report

Produces a rules report. See the "Rules Report" topic in this chapter for more information.

Class Rules

Use the Class Rules dialog box to define rules that apply to a collection of nets known as a *net class* and to multiple net classes.

1. Choose Setup/Design Rules. The Rules dialog box appears.
2. Choose the Class icon. The Class Rules dialog box appears.
3. Select the net classes you want to define Clearance, Routing, and High-speed rules for and choose the appropriate icon.



Class Name

Specifies the new or selected class name.

Class

Lists defined net classes by name and parenthetically notes the rules that apply, if any, to the class.

Add

Defines a class. Type a name in the Class Name text box and choose Add.

Delete

Removes the selected class from the list. Choose a name in the list box and choose Delete.

Rename

Modifies the class name. Edit the name in the Class Name text box and choose Rename.

Net Class Rules Icons

Defines Clearance, Routing, or High-speed rules for a net class. Choose the appropriate icon.

Clearance	Opens the Clearance Rules dialog box where you set net class Clearance rules. See the "Clearance Rules" topic in this chapter for more information.
Routing	Opens the Routing Rules dialog box where you set net class Routing rules. See the "Routing Rules" topic in this chapter for more information.
High-speed	Opens the High-speed Rules dialog box where you set net class High-Speed rules. See the "High-speed Rules" topic in this chapter for more information.

When you select a class name, an icon appears below each rule type to indicate the hierarchy level where the rule is defined for that class. For example, a class with only Clearance rules defined has a class icon below the Clearance icon and Default icons below the Routing and High-Speed icons.

Report

Produces a Class rules report. See the "Rules Report" topic in this chapter for more information.

Show Classes with Rules

Lists only classes with at least one set of rule definitions.

Nets Area

Adds or removes selected nets from the net class.

Choose a class in the list box to display nets in the Available and Selected list boxes, or choose the Add button to create a new class. To assign nets to a class, choose the nets in the Available list box and choose Add.

Note: A net can only belong to one net class. The Available list box only contains nets that do not belong to a net class.

To delete nets from the class, choose the nets in the Selected list box and choose Remove.

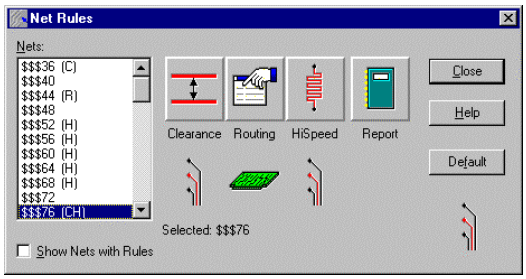
Default

Returns the class rules to the default rules.

Net Rules

Use the Net Rules dialog box to define rules that apply to a single net or multiple nets.

1. Choose Setup/Design Rules. The Rules dialog box appears.
2. Choose the Net icon. The Net Rules dialog box appears.
3. Select the nets you want to define Clearance, Routing, and High-Speed rules for and choose the appropriate icon.



Nets

Lists nets by name and parenthetically notes any rules that apply to the net. Select nets and choose the appropriate rule icon to apply rules.

Net Rules Icons

Defines Clearance, Routing, or High-speed rules for nets. Choose the appropriate icon.

Clearance	Opens the Clearance Rules dialog box where you set Clearance rules for a net. See the "Clearance Rules" topic in this chapter for more information.
Routing	Opens the Routing Rules dialog box where you set Routing rules for a net. See the "Routing Rules" topic in this chapter for more information.
High-speed	Opens the High-speed Rules dialog box where you set High-speed rules for a net. See the "High-speed Rules" topic in this chapter for more information.

When you select a net name, an icon appears below each rule type to indicate the hierarchy level where the rule is defined for that net. For example, a net with both Clearance and High-Speed rules defined has net icons below the Clearance and High-speed icons and a Default icon below the Routing icon.

Report

Produces a Net rules report. See the "Rules Report" topic in this chapter for more information.

Show Nets with Rules

Lists only nets with at least one set of rule definitions.

Default

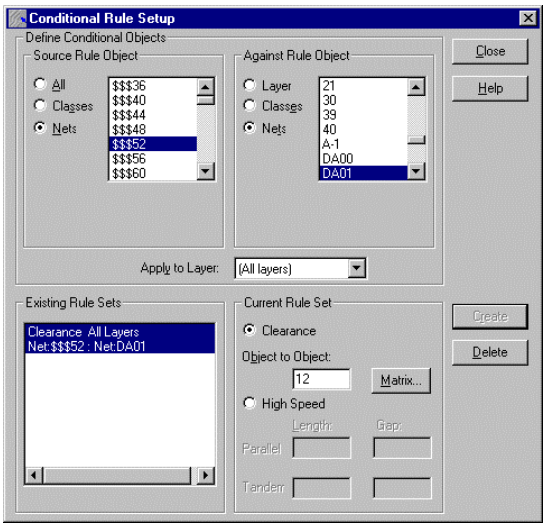
Returns the net rules to the default rules.

Conditional Rules

Once you set up Clearance rules for a group in the hierarchical order, the rules are applied to all other objects. Use the Conditional Rules Setup dialog box to apply a third overriding set of rules that apply only when the item meets other specific levels of the hierarchical order.

You can use a layer as an *against object*, where rules you set for an object like a net apply only when the net is routed on that layer. You can further refine this to use another net as an against object and specify a layer to which the rules to apply. If these two nets meet on this layer, they must adhere to this clearance. You define these relationships by making conditional rule sets.

1. Choose Setup/Design Rules. The Rules dialog box appears.
2. Choose the Conditional Rules icon. The Conditional Rules Setup dialog box appears.
3. Identify the items in the source and against list boxes.
4. Set the option under Current Rule Set to Clearance or High-speed.
5. Choose Create.
6. Select the rule set in the Existing Rules Sets area and define the appropriate values for the rule set.



Source Rule Object Area

Establishes all objects, or a selected class or net, as the subject of the rule.

Note: The lower the source object's place in the hierarchical order, the broader the selection of items that you can select from in the Against list box. This helps you create rule sets in one direction, and avoid conflicting hierarchical items.

Against Rule Object Area

Identifies all objects, or a selected class or net, the rule set is checked against.

Apply to Layer Area

Uses a layer as an against object or applies an item-to-item rule on a specific layer.

Existing Rule Sets Area

Lists defined rule sets for specific Clearance or High-speed definition of the selected rule set.

Clearance Rule Set Area

Sets Clearance rules for the rule set.

Once you specify a Clearance rule set, you can enter the minimum clearance gap you want the source and against items to maintain from each other. Choose Matrix to enter more item-specific standoffs.

High-speed Rule Set

Specifies High-speed rules for the rule set.

High-speed lets you enter clearance values for parallel and tandem length checking for the condition, and it uses your source-against entries as the victim-aggressor designations for cross-talk conditions checking.

Once you specify a high-speed rule set, you can set specific maximum length and minimum gap for Parallel and Tandem checks. These rules apply only to the source and against object when they meet each other. For parallelism reporting purposes, the source object is treated as the aggressor. Whatever clearances are set for the source object, local or default, appear in the Values boxes as a reference. If you specify a class as the source, all nets in the class are checked as victims.

Create

Compiles the new rule set parameters and adds a description to the Existing Rule Sets list box. After you create a rule set, define the Clearance or High-Speed values.

Delete

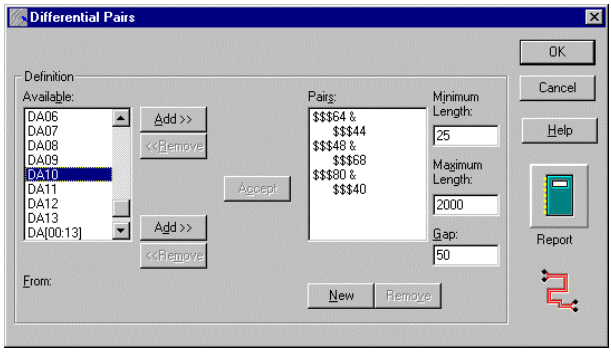
Removes the selected rule set from the Existing Rule Sets list box.

Differential Pairs

Use the Differential Pairs dialog box to associate two nets and define rules related to the pair.

1. Choose Setup/Design Rules. The Rules dialog box appears.
2. Choose the Differential Pairs icon. The Differential Pairs dialog box appears.

Note: These rules are not checked; they are mainly high-level autorouter setups.



Available

Lists the nets that are available for pairing.

To pair nets, choose a net in the list box and choose either the upper or lower Add button. Choose another net and choose the other Add button. Choose Remove to select a different net during the pair definition.

Note: A net can only belong to one net pair. The Available list box only contains nets that do not belong to a differential pair.

Pairs

Lists the net pairs in the list box.

Length and Gap

Specifies or modifies the minimum and maximum trace length and gap between the traces. Choose Accept when your definition is complete.

New

Begins the definition process for pairing net.

Remove

Disbands a differential net pair and removes it from the Pairs list.

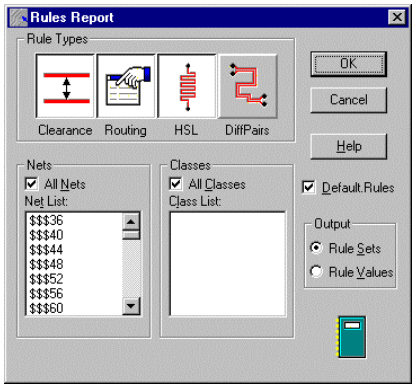
Report

Produces a report on Differential Pairs. See the "Rules Report" topic in this chapter for more information.

Rules Report

Use the Rules Report dialog box to produce a report of some or all of the rules you defined. By default, you produce a complete report of all rules.

1. Choose Setup/Design Rules. The Rules dialog box appears.
2. Choose the Report icon. The Rules Report dialog box appears.



Rule Types Area

Selects the rules to report. Choose any combination of icons, including Differential Pairs to report net pairs.

Nets Area

Reports the specified rules for every net or selected nets. Choose All Nets or choose specific nets.

Classes Area

Reports the class rules for every selected rule set. Choose All Classes or choose specific net classes.

Default Rules

Reports the default rules for the specified nets and classes.

Output Area

Reports all rules in the current hierarchy that are unique from the default (Rule Sets) or reports the values of all rules in the current hierarchy level even if they match the default (Rule Values).

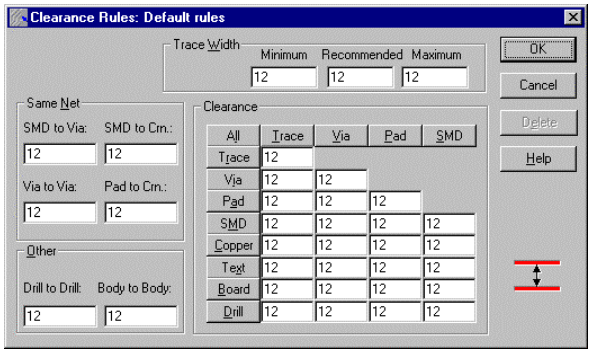
Clearance Rules

Use the Clearance Rules dialog box to define the legal spacing between objects. When imported into PowerPCB, the On-line DRC and Verify Design programs use these rules to check and report clearance violations.

You can use the Conditional Rules dialog box to save a clearance configuration as a set, to apply for a selected item only when it comes in contact with different level of the hierarchical order.

To open the Clearance Rules dialog box:

1. Choose Setup/Design Rules. The Rules dialog box appears.
2. Choose the Default, Class, or Net icon. The appropriate dialog box appears.
3. Choose the Clearance icon. The Clearance Rules dialog box appears. A dialog box subtitle states the set of rules for which you are setting Clearance rules, Default, Class, or Net.



Trace Width

Restricts the trace width to a range of values. The Recommended value is the width you want to assign to the trace when routing begins. The Minimum and Maximum values are respected by routing routines, which must use trace width to achieve some high-speed routing functions, like impedance matching.

Same Net Area

Applies edge-to-edge clearance values to items which are in the same net.

SMD to Via	Minimum spacing between a surface mount pad and escape via.
SMD to Corner	Minimum spacing between a surface mount pad and the first trace bend point.
Via to Via	Minimum spacing between two vias in the same net.
Pad to Corner	Minimum spacing between a through hole pad and the first trace bend point.

Other Clearance Values Area

Sets other clearance values.

Drill to Drill	The minimum edge-to-edge spacing between two drill holes.
Body to Body	The minimum edge-to-edge spacing between two component bodies.

Clearance Matrix Area

Defines edge-to-edge clearances between two object types.

To define the minimum spacing between any two objects, type the value in the appropriate text box.

To define the same spacing value for all text boxes in one matrix column, press the button above the column. Type a value and choose OK.

To define the same spacing value for all text boxes in one matrix row, press the button in the left column. Type a value and choose OK.

To define the same spacing value for all text boxes in the matrix, press the All button. Type a value and choose OK.

Delete

Removes a set of Clearance rules from your rules hierarchy.

You cannot delete the Default Clearance rules.

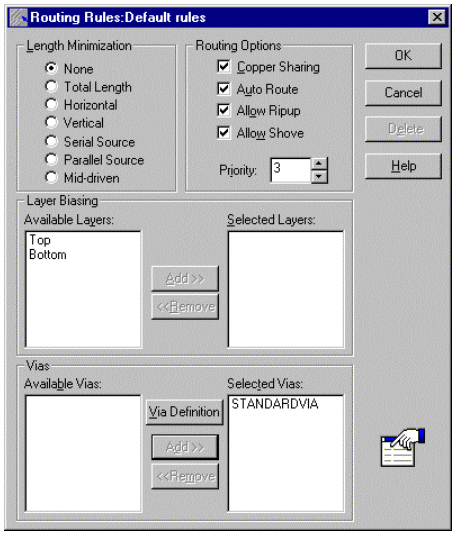
Routing Rules

Use the Routing Rules dialog box to define how to interactively and automatically connect nets.

To open the Routing Rules dialog box:

1. Choose Setup/Design Rules. The Rules dialog box appears.
2. Choose the Default, Class, or Net icon. The appropriate dialog box appears.

3. Choose the Routing icon. The Routing Rules dialog box appears. A dialog box subtitle states the set of rules for which you are setting Routing rules, Default, Class, or Net.



Length Minimization Area

Defines how to reproduce the *ratsnest*, which is made of lines defining the pin-to-pin connections, as parts are moved. Choose a Length Minimization setting:

None	No automatic regeneration.
Total	Minimizes total length of all nets.
Horizontal	Minimizes horizontal length of all nets.
Vertical	Minimizes vertical length of all nets.
Serial Source	Places multiple source pins in series. Used with ECL nets.
Parallel Source	Places multiple source pins in parallel. Used with ECL nets.
Mid-driven	Minimizes the distance between all pins in a net and organizes the connections giving a specific net order. Mid-driven tries to make the net as short as possible. Terminators are placed at each end of the net and the loads are evenly distributed on each branch. There must be exactly two terminators or the net is ordered as a simple, optimized daisy chain. Sources are placed in the middle of the net, and if there is more than one source, the sources are chained together before the rest of the net is processed. This command is usually used for High-Speed requirements. It is also used with ECL nets.

Routing Options Area

Sets routing options.

Copper Sharing	Two pin pairs can share copper. Also known as t-routing.
Auto Route	An autorouter can route the net.
Allow Ripup	An autorouter can automatically rip up and reroute a routed net. Interactive routing checks this setting. You can rip up traces when you edit them if this check box is on. You must turn this check box on if you want to unroute traces while using DRC Warn or Prevent.
Allow Shove	An autorouter can move a routed net. Interactive routing checks this setting. You can shove traces when you edit them if this check box is on. You must turn this check box on if you want to shove traces while using DRC Warn or Prevent.
Priority	The priority an autorouter uses. Specify a value from 0 to 100. Nets with higher priority are routed first.

Layer Biasing Area

Restricts nets to certain layers for routing.

1. Select the layers you want to restrict nets to in the Available Layers list box.
2. Choose Add to add them to the Selected Layers list box.

To unrestrict layers:

1. Select the layers you want unrestricted in the Selected Layers list box.
2. Choose Remove to move them back to the Available Layers list box.

Nets are restricted to layers specified in the Selected Layers list box. If the Available Layers list box is empty, the net has no layer restrictions.

Vias Area

Restricts which via types to use.

1. Select the vias you want to use in the Available Vias box.
2. Choose Add to add them to the Selected Vias list box.

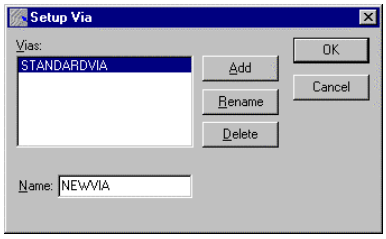
To unrestrict via types:

1. Select vias in the Selected Vias list box.
2. Choose Remove to move them back to the Available Vias list box.

Via use for the net, class, etc. is restricted to the via types in the Selected Vias list box. If the Available Vias list box is empty, there are no via restrictions.

Choose Via Definition to create, rename, or delete a via in the Setup Via dialog box.

Setup Via Dialog Box



Note: If you define new via types in your schematic, you must also define the via types in PowerPCB using padstack setup before reading the netlist into PowerPCB.

Add

Adds a new via type to the schematic. You must name the new via type.

Rename

Renames an existing via type in the schematic.

Delete

Removes a via type from the schematic. You cannot remove standard vias.

Name

Names a new via type. Type the name for a new via type, then choose Add.

High-speed Rules

Use the High-speed Rules dialog box to define rules for the following parameters:

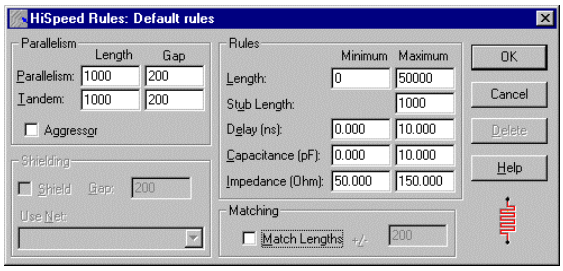
- Parallel
- Tandem
- Impedance
- Capacitance
- Routed length
- Stub length
- Shielding
- Matched length

When imported into PowerPCB, the EDC (Electrodynamic Checking) routine checks to see if rules are met correctly after routing (except shielding and matched length).

You can use the Conditional Rules dialog box to save a high-speed configuration as a set, to apply for a selected item only when it comes in contact with different level of the hierarchical order.

To open the High-speed Rules dialog box:

1. Choose Setup/Design Rules. The Rules dialog box appears.
2. Choose the Default, Class, or Net icon. The appropriate dialog box appears.
3. Choose the High-speed icon. The High-speed Rules dialog box appears. A dialog box subtitle states what set of rules you are setting High-speed rules for: Default, Class, or Net.



Parallelism Area

Specifies rules for parallel and tandem checking. This value determines the distance and standoff against the specific item and everything else in the design. To set a narrower check, use Conditional Rules to define specific Source and Against items.

Parallel Rules	Restricts the distance that traces in *different nets* on the *same layer* can run together.
Tandem Rules	Restricts the distance that traces in *different nets* on *different layers* can run together.

Define the following parallel/tandem rules:

Length	Sets the maximum parallel/tandem distance.
Gap	Sets the minimum gap between traces below which the parallel/tandem rules apply.

Choose Aggressor to specify if a net is an *aggressor*, or source of interference, during parallel/tandem checks.

Shielding Area

Invokes the arrangement of certain nets to act as shields for others; for example the net in Use Net is routed up and down both sides of a selected net to provide protection from interference.

Note: You can only assign nets associated with plane layers in the Layer Definition dialog box to shield other nets. If no plane layers exist, the Shielding area grays.

If your router supports shielding, you can specify the shield gap value and net to use as the shield in the Gap and Use Net text boxes.

Although you can create these rule types in PowerLogic and pass them to PowerPCB, PowerPCB does not check against these values.

Rules Area

Sets the verify design/high-speed checking routine that PowerPCB uses.

Length	Sets a minimum and maximum length.
Stub Length	Sets a maximum stub length. The stub length is the distance from a T-point to the end of the route.
Delay	Sets a minimum and maximum delay time in nanoseconds.
Capacitance	Sets a minimum and maximum capacitance in picofarads.
Impedance	Sets a minimum and maximum impedance in ohms.

Matching

Turns length matching on. Length matching is a *same length requirement* parameter you can pass on to autorouters which support it. The value represents the plus or minus tolerance inside which the router determines lengths for the specified nets. Length matching rules are usually set for all nets in a class.

Although you can create these rule types in PowerLogic and pass them to PowerPCB, PowerPCB does not create items for, or specifically check against, these values.

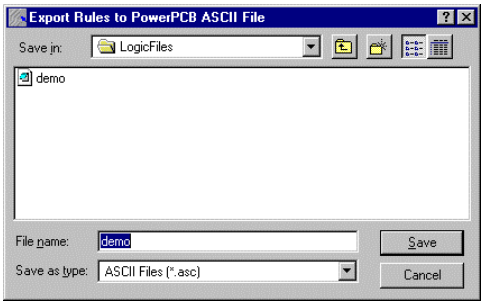
Delete

Removes this set of High-speed rules from your rules hierarchy. You cannot delete the default High-speed rules.

Export Rules to PCB

Use Tools/Export Rules to PCB to write design rules to an ASCII file, including layer setups and layer counts, etc.

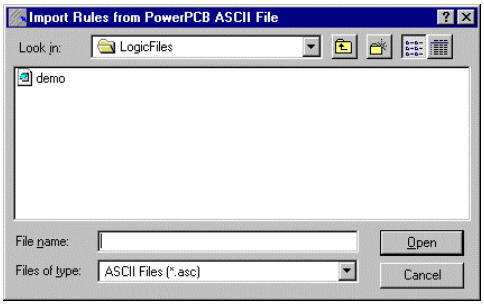
1. Choose Tools/Export Rules to PCB. The Export Rules to PCB ASCII File dialog box appears.
2. Type a new filename or choose an existing file.
3. Choose Save to create the file.



Import Rules from PCB

Use Tools/Import Rules from PCB to read design rules from an ASCII file, including layer setups and layer counts, etc.

1. Choose Tools/Import Rules from PCB. The Import Rules from PCB ASCII File dialog box appears.
2. Type a new filename or choose an existing file.
3. Choose Open to read the file.

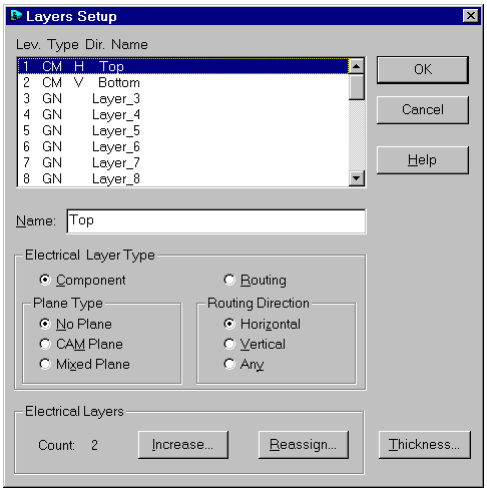


Setting Layers

If you plan to netlist to PowerPCB, use the Layers Setup dialog box to define electrical layers and associate component and documentation layers. Choose Setup/Layer Definition to open the dialog box.

The first lines are reserved for electrical layers. Electrical layers are all layers between, and including, the top and bottom layers. You can add and edit traces and vias on these layers. To add the inner electrical layers, use the Increase button in the Electrical Layers area.

Layers below the bottom layer on the list are nonelectrical or documentation layers.



List Area

Displays the layer name, type, and routing direction for each assigned layer of the design in a list box.

Name

Specifies a layer name that defines its function. Type a descriptive layer name in the text box.

Electrical Layer Type

The Top and Bottom layers can be component, routing, plane, or split/mixed plane layers. Inner layers can be routing, plane, or split/mixed plane. When a layer is selected as a plane or split/mixed plane layer, use the Assign Nets icon to assign which nets to connected to it.

Electrical Layers Area

Lets you to increase the number of electrical layers and reassign data from one layer to another. Choose the Increase or Reassign button.

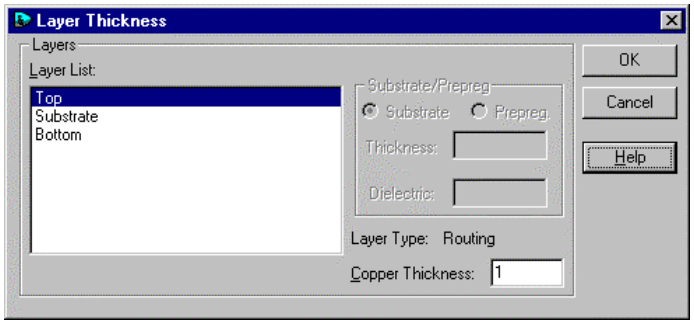
Thickness

Opens the Layer Thickness dialog box where you can define layer and substrate thickness and dielectric information. The Electrodynamic Checking module, EDC, uses this information.

Layer Thickness Dialog Box

Use the Layer Thickness dialog box to define layer and substrate thickness and dielectric information. The Electrodynamic Checking module, EDC, uses this information. To open this dialog box:

1. Choose Setup/Layers.
2. Choose the Thickness button. The electrical layers plus the substrate/prepreg layers between the electrical layers appear in the list box.



Layer List

Lists the layers for which you can define substrate, prepreg, or copper laminate thickness.

Layer Type

The Layer Type for the selected copper laminate layer is displayed here.

Copper Thickness

Sets the Copper Thickness on the selected copper laminate layer. This option grays when a substrate layer is selected.

Substrate/Prepreg Area

These options are available when you choose a substrate layer from the Layer List. This area grays when a copper laminate layer is selected.

Substrate or Prepreg
: Selects a substrate laminate pair or a prepreg sheet for the selected substrate layer.

Thickness
: Sets the Thickness of the selected substrate layer.

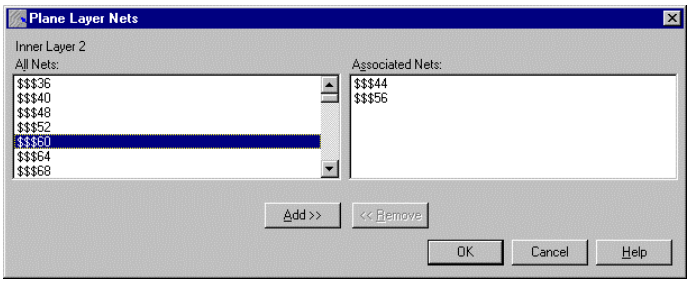
Dielectric Constant
: Sets the electrical characteristics for the selected substrate layer.

Plane Layer Net Assignments

Plane layers contain large copper areas which components can access for ground and power. PowerPCB produces thermal relief connection pads for nets associated with the plane layer.

Use the Layers Setup dialog box to define a layer as a plane layer and assign one or more net names to a plane layer. You can also assign a net to more than one plane layer.

1. Choose Setup/Layer Definition. The Layers Setup dialog box appears.
2. Choose the layer you want to define as a plane layer in the Setup Layers dialog box. It must be an electrical layer, between top and bottom. Choose Increase to add electrical layers.
3. Choose the Plane button. The Assign Nets icon appears.
4. Choose the Assign Nets icon. The Plane Layer Nets dialog box appears.



5. Choose the net name(s) to add in the All Nets list box.
6. Choose Add. The net names appear in the Associated Nets list box. Choose Remove to unassign net names.

Increasing the Number of Electrical Layers

To increase the number of electrical layers in the database:

1. Choose Setup/Layer Definition. The Layers Setup dialog box appears.
2. Choose Increase.
3. You are prompted for the new number of electrical layers. Type a value between the current number of electrical layers and 30.

   You cannot decrease the number of electrical layers.

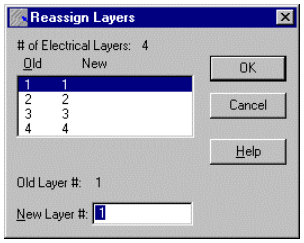
   The Reassign Layers dialog box appears. You can reassign the electrical information on any existing electrical layer to a new layer.

   The new electrical layers enter the database with default parameter values.

Reassigning Electrical Layer Data

Use the Reassigning Layers dialog box to move data from one electrical layer to another in PowerPCB.

1. Choose Setup/Layer Definition. The Setup Layers dialog box appears.
2. Choose Reassign. The Layer Reassignment dialog box appears. By default, there is no reassignment.



3. Select the list box row corresponding to the old layer number.
4. Type the new layer number in the text box. The following information moves from the old layer to the new layer:
   - Traces and vias.
   - Drafting objects.
   - Layer name, layer type, routing direction, and component/plane layer parameters.
5. Choose OK.

Chapter 11
Printing and Plotting

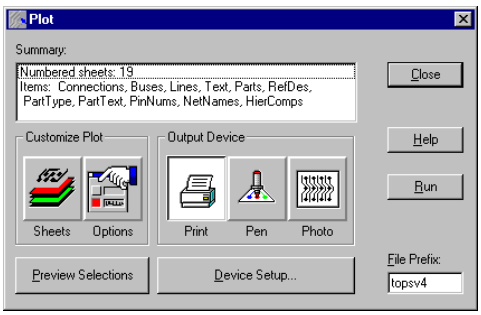
This chapter discusses the different outputs you can create from your schematic in PowerLogic. You can print, plot, and create reports of your schematic. This chapter includes information on setting up your printing or plotting device, previewing the output, and creating the output. Most topics in this chapter are related to both printing and plotting, except where noted. The topics in this chpater include:

Print/Plot Operations
Creating Reports

Print/Plot Operations

Use the Print/Plot command on the File menu to setup and print your schematic. To print or plot:

1. Choose File/Print/Plot. The Plot dialog box appears. Use the Print/Plot dialog box to produce laser printouts, line plots, and photoplot outputs.
2. Select the sheets and the items to plot by choosing the Sheets icon. See the "Select Items" topic in this chapter for more information.
3. Choose Preview Selections to preview the layer and items selections you made.
4. Define plot options by choosing the Options icon. See the "Plot Options" topic in this chapter for more information.
5. Set up the output device you want to use by choosing the icon for the device and choosing Device Setup. Choose from Printer, Pen Plotter, or Photoplotter.



Summary

Displays the sheet selection settings.

Customize Plot Area

Tailors the print characteristics of the schematic (for example, which sheets to print, which items to print, how to orient the plot, etc.). Double-click on the Sheets or Options icon to open the Select Items or Plot Options dialog box. See the "Select Items" topic in this chapter for more information.

Output Device Area

Specifies the device type for the set up or preview functions. Choose the Print, Pen, or Photo icon before choosing either Preview Selections or Device Setup.

Preview Selections

Displays the selected print settings in the Selections Preview dialog box. The preview display corresponds to the selected output device.

Device Setup

Invokes a set up dialog box for the selected output device.

Run

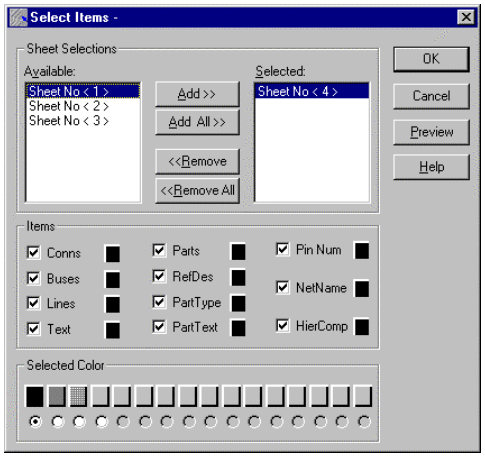
Sends the print/plot request to the designated printer/plotter or file.

File Prefix

Specifies the plot filename prefix for either a pen plot or photoplot print to file function.

Select Items

Use the Select Items dialog box to specify which sheets and schematic items to print /plot. To open this dialog box, choose Sheets in the Customize Plot area of the Print/Plot dialog box.



Available

Lists all available sheets. Double-click on the item name to add it to the Selected list.

Selected

Lists all sheets selected for plotting.

Add

Adds the selected sheet(s) in the Available list to the Selected list.

Add All

Adds every sheet in the Available list to the Selected list.

Remove

Removes the selected sheet(s) from the Selected list and returns them to the Available list.

Remove All

Removes every sheet from the Selected list and returns them to the Available list.

Items Area

Specifies items for printing/plotting and assigns colors to items. Items with a check box that is turned on will print/plot. To assign color to an item, first choose a color in the Selected Color area, then choose the color box to the right of each item that you want to change.

Selected Color Area

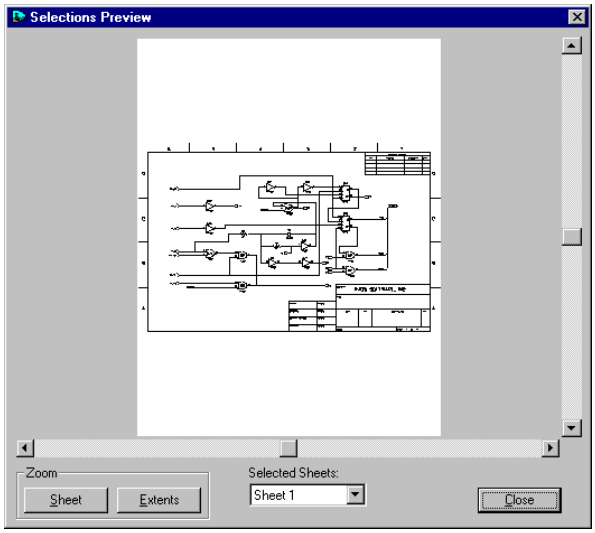
Specifies a color for assignment to items.

- For pen plots and color printers, choose one of the available colors. To change the available colors, refer to the "Pen Plotter Setup" topic in this chapter.
- For black and white printers and photoplots, black is the only available color.
- For PostScript output, select a gray-scale entry.

Preview

Displays the selected sheets and items in the Selections Preview dialog box. Use the Selections Preview dialog box to graphically verify the print/plot settings. To open this dialog box, either choose Preview in the Select Items dialog box or choose Preview Selections in the Print/Plot dialog box.

Click the left mouse button in the Preview window to zoom in at the cursor location. Click the right mouse button to zoom out.



Sheet

Displays only the sheet.

Extents

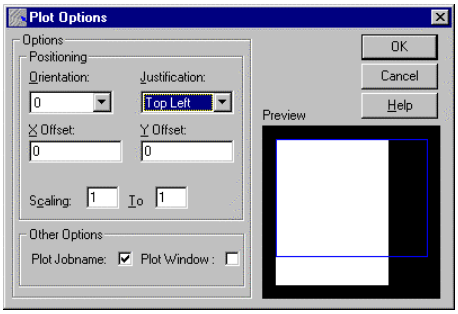
Displays the sheet and the paper dimensions.

Selected Sheets

Contains a list of sheets. Choose which sheet to display from the drop-down list box.

Plot Options

Use the Plot Options dialog box to define the options associated with an individual plot. To open this dialog box, choose Options in the Customize Plot area of the Print/Plot dialog box.



Orientation

Sets the orientation angle.

Justification

Positions the plot. Choose from: Centered, Bottom Left, Bottom Right, Top Left, Top Right, or Offset. For any choice but Scale to Fit or Centered, you can define the margin offsets.

X and Y Offsets

Sets offset values. You can only set offset values when Justification is a position other than Scale to Fit or Centered.

Note: The plot location is calculated after the plot is rotated and scaled.

Scaling

Defines the plot size to actual size ratio. 2 to 1 scaling results in a plot that is twice real size.

Plot Job Name

Plots the schematic name, time, and date.

Plot Window

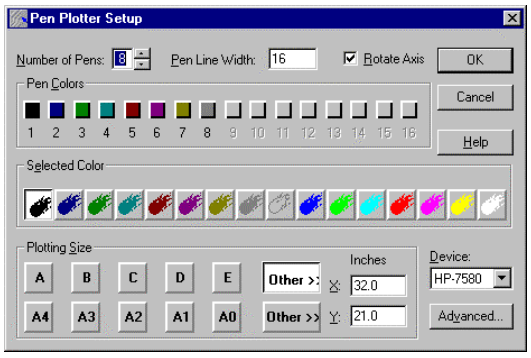
Plots the current view of the schematic window.

Preview

Reflects changes you make to the Positioning options to show how the information will plot. The plotted information is represented by a blue rectangle.

Pen Plotter Setup

Use the Pen Plotter Setup dialog box to set pen colors and other global pen plot options. To open this dialog box, first choose Pen in the Output Device area of the Print/Plot dialog box and then choose Device Setup.



Number of Pens

Specifies the number of pens in your device (1-16).

Pen Line Width

Sets the pen width in mils.

Rotate Axis

Reverses the X and Y axis.

Pen Colors Area

Assigns colors to pens. To assign color to a pen, first choose a color in the Selected Color area, and then choose the color box above each pen number that you want to change.

Selected Color Area

Specifies a color assignment for pens.

Plotting Size Area

Sets the plot size. Choose Other to define a custom plot size; type the X and Y dimensions.

Device

Lists the available pen plotter devices in a drop-down list box.

Advanced

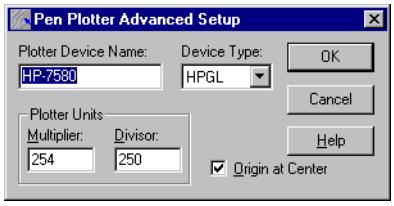
Opens the Pen Plotter Advanced Setup dialog box where you can specify advanced pen plotter settings. If your device type is not listed, choose Advanced to define its characteristics and add it to the device list.

Pen Plotter Advanced Setup

Use the Pen Plotter Advanced Setup dialog box to add a new pen plotter to the list of available plotters. To open this dialog box, choose Advanced from the Pen Plotter Setup dialog box.

You can also edit the data for an existing plotter, but only if it is a custom setup. You cannot modify the PADS-supplied advanced plotter settings.

The new plotter must support either the HPGL or HGML interface language. Other plotter languages or formats are not yet supported.



Plotter Device Name

Specifies the model number of the plotter that you want to add. Do not reuse one of the existing PADS-supplied plotter device names.

Device Type

Specifies the plotter interface language, either HPGL or HGML.

Plotter Units Area

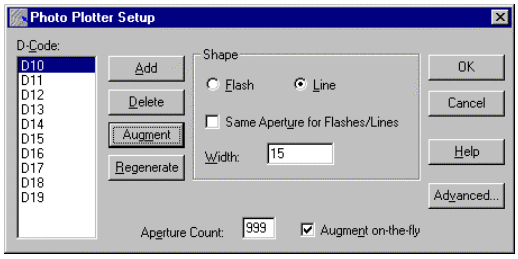
Defines the plotter resolution with a scaling ratio that you form by specifying both a Multiplier and Divisor value. The ratio defined is the scale factor to convert from mils (0.001 in) to plotter units. For example, most Hewlett-Packard plotters have a resolution of 0.025 mm or 1/40 mm. This means that a distance of one inch (1000 mils) is 1016 plotter units (25.4 X 40). So a ratio of 1016 to 1000 would be defined. The ratio actually used is 254 to 250 which is the same as 1016 to 1000.

Origin at Center

Sets the origin at the center of the paper. Most plotters have their plotter origin either in the center of the paper or at the lower left corner. Ensure that this check box is on if the origin is in the center.

Photo Plotter Setup

Use the Photo Plotter Setup dialog box to define the apertures and other photoplot options. To open this dialog box, first choose Photo in the Output Device area of the Print/Plot dialog box and then choose Device Setup.



D-Code

Lists all defined apertures.

Add

Adds a new aperture to the list. A prompt asks for the new D-code number.

Delete

Deletes an aperture from the list. Select the aperture and choose the Delete button. Confirm the deletion.

Augment

Automatically adds D-codes for items in the schematic file that are not in the current list.

Regenerate

Clears the current list and automatically adds D-codes for all items in the schematic file.

Shape Area

Specifies a shape for a D-code. Choose a D-code and type values in the text boxes to define the following:

Flash	Indicates a flash aperture.
Line	Indicates a paint aperture.
Same Aperture for	Flashes/Lines Draws lines and flashed items with the same aperture. Round and square shapes for lines gray.
Width	Specifies the width, or the diameter for round shapes. The text box grays if the value is not appropriate for the specified shape.

Aperture Count

Defines the maximum aperture count; type a value in the text box.

Augment On-the-fly

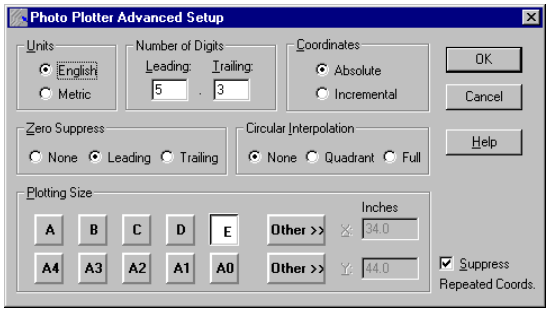
Automatically adds apertures to the list as you create the lines that use them. Necessary apertures always appear in the Photo Plotter Setup dialog box.

Advanced

Opens the Photo Plotter Advanced Setup dialog box where you can define additional photoplotter global parameters.

Photo Plotter Advanced Setup

Use the Photo Plotter Advanced Setup dialog box to set global photoplotter options. To open the dialog box, choose Advanced in the Photo Plotter Setup dialog box.



Units Area

Defines the measurement value.

English	For mils.
Metric	For millimeters.

Number of Digits Area

Defines the output file coordinate precision. You can specify the number of digits before, Leading, and after, Trailing, the decimal point.

Coordinates Area

Defines the output file coordinate type.

Absolute	Specifies absolute coordinates.
Incremental	Specifies relative coordinates.

Zero Suppress Area

Defines the output file coordinate format.

None	Retains both leading and trailing zeros.
Leading	Suppresses leading zeros.
Trailing	Suppresses trailing zeros.

Circular Interpolation Area

Defines how to draw arcs and circles.

None	Choose None if your photoplotter does not support circular interpolation to draw arcs and circles as small straight-line segments.
Quadrant	Choose Quadrant if your photoplotter does not support full 360-degree circular interpolation.
Full	Choose Full if your photoplotter supports full 360-degree circular interpolation.

Plotting Sizes Area

Lists plotting sizes. Choose the icon for the plotting size you want to use. If you choose Other, define a paper size by X and Y dimensions in inches.

Suppress Repeated Coordinates

Eliminates repeated coordinates from the output file.

PostScript™ Printing to a File

PowerLogic supports PostScript printing to a file using the Windows Print Manager. For information on using the Print Manager, refer to the *Microsoft Windows User's Guide*. To create a PostScript file, set the printer to print to a file in the Print Manager.

Setting the Printer to Print to a File Using the Print Manager

In Windows 95 and Windows NT 4.0:

1. Choose Start/ Settings/Printers.
2. Select the PostScript printer you want to use to print to a file.
3. Choose File/Properties.
4. For Windows 95, choose the Details tab in the Properties dialog box.

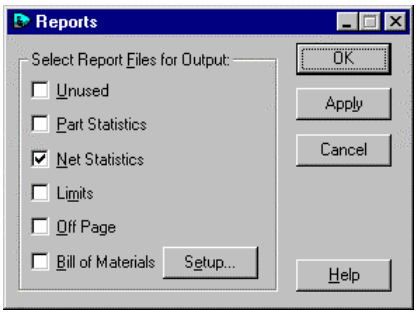
   For Window NT 4.0, choose the Ports tab in the Properties dialog box.
5. For Windows 95, choose FILE in the Print to the following port drop-down list box.

   For Windows NT 4.0, choose FILE as the port option.
6. Choose OK. You are now set up to print to a file.

Creating Reports

Use the Reports dialog box to produce any of six different types of reports on the current schematic. You can save these reports as text files on your hard disk or output them to a printer.

1. Choose Report from the File menu. The Reports dialog box appears.
2. Select the check box on for each report you want to generate. PowerLogic directs all reports, except the netlist report, padsnet.asc, into a single text file, report.rep.
3. If you choose Bill of Materials Setup, you can specify or modify the report characteristics.
4. When you finish report selection, and the Bill of Materials setup, choose OK. PowerLogic generates the reports and the dialog box closes. Choose Cancel to close the dialog box without generating reports.



Unused

Lists all unused pins and gates in the schematic.

Part Statistics

Lists all parts in the schematic.

Net Statistics

Lists all nets in the schematic and identifies any questionable nets.

Limits

Compares the current schematic to the maximum schematic size allowed.

Off Page

Lists all signals that appear on more than one page, with sheet number and X,Y location.

Bill of Materials

Produces a bill of materials in a user-customizable format.

Unused

The Unused report lists all unused gates and pins in the schematic. Use this report for troubleshooting and to maximize part usage. The Unused report first lists unused parts and gates, followed by a list of unused pins on each part.

The format for the Unused report follows:

```
UNUSED ITEMS REPORT – \PADSPWR\LOGICFILES\filename.SCH –date time
UNUSED GATE LIST
part type
gate name    gate name
part type
gate name    gate name
UNUSED PINS LIST
part type
ref des. pin number ref des.pin number
```

Part Statistics

The Part Statistics report lists information about all parts in the schematic. The report includes the reference designator name and part type for each part in the schematic and for each pin on the part, the pin type, sheet location, and signal name. Use this report to locate possible errors in the schematic.

The format for the Part Statistics report follows:

```
PART STATUS REPORT – \PADSPWR\LOGICFILES\filename.SCH –date time
Ref name     Part type
ref.pin no. [pin type - sheet no.] signal name
ref.pin no. [pin type - sheet no.] signal name
ref.pin no. [pin type - sheet no.] signal name
Ref name     Part type
ref.pin no. [pin type - sheet no.] signal name
ref.pin no. [pin type - sheet no.] signal name
```

Net Statistics

The Net Statistics report lists information for each net in the schematic. The report includes the reference designator name and pin number for all parts in the net. PowerLogic flags possible errors (for example, nets with no inputs or no outputs, nets with multiple outputs, etc.) for further examination. Use this report to locate possible errors in the schematic.

The format for the Net Statistics Report follows:

```
NET STATUS REPORT — \PADSPWR\LOGICFILES\filename.SCH —date time
net name      SOURCES = n   LOADS = m    UNDEFINED = x
ref des. pinnumber [pin type - sheet no.]
ref des. pinnumber [pin type - sheet no.]
```

Error Messages

The following error messages can occur:

Net has only one pin: a net going to an off-page flag not connected elsewhere.

Net has no defined source: there is no pin in the net that has a pin type of S.

Net has no defined loads: there is no pin in the net that has a pin type of L.

Net has multiple sources- be sure they are or-tieable: the net has more than 1 pin with the pin type S.

Job Limits

The Job Limits report indicates the maximum number of each PowerLogic data item (parts, nets, text) your system will allow, as well as the current count of each of these items in the schematic. This limit varies depending on the amount of virtual memory that is available.

The report has two parts. The first is a list of items whose limits are common to the entire schematic. The second is a count, for each schematic sheet, of the items whose limits apply for each sheet.

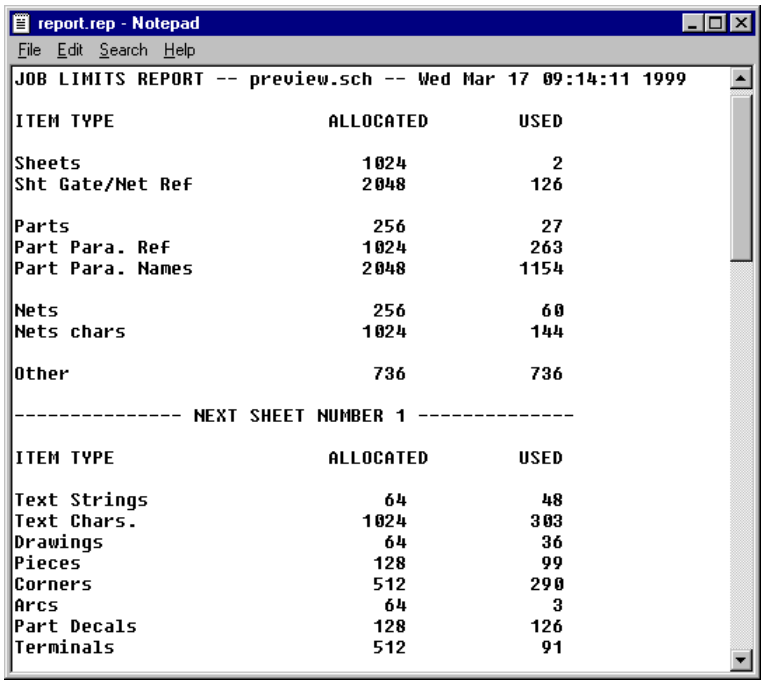
You should periodically run a Limits report to ensure that you are not approaching the system's limit for any item.

If you exceed the Maximum No. of Items for any item, you cannot continue adding those items to the schematic. The solution is to split the design into multiple schematics, run separate netlists for each schematic, then merge the netlists using a text editor.

An example of the Limits report follows:

```
JOB LIMITS REPORT — \PADSPWR\LOGICFILES\filename.SCH — date time
ITEM TYPE            ALLOCATED   USED
Sheets               1024  1
Sht Gate/Net Ref     32767 15
Parts                5120  2
Part Para. Ref       102400      16
Part Para. Names     65536 112
Nets                 10240 5
Nets chars           32767 0
Other                1008  2

       ————————NEXT SHEET NUMBER 1 —————————
ITEM TYPE            ALLOCATED   USED
Text Strings         4096        0
Text Chars.          32767       0
Drawings             4096        7
Pieces               8192        45
Corners              32767       107
Arcs                 2048        0
Part Decals          1024        24
Terminals            16384       8
Part Types           2048        1
Gates                4096        6
Gate Pins            32767       12
Gate Sig Pins        4096        0
Pin Num Names        12288       0
Pin Names            32767       0
Gates/Comps.         8192        7
Comp pins            32767       14
Nets names           4096        0
Tie Dots             4096        3
Off Page Refs        4096        0
Busses               512         0
Connections          16384       9
Route Segments       32767       22
Other                20          20
```



Off Page

The Off Page report lists the X,Y coordinate location and sheet number of all off-page, ground, and power symbols in the schematic. Use the report to quickly locate an off-page symbol using the S (Search) modeless command.

An error message appears when a net contains only one off-page reference.

The format of the Off Page report follows:

```
OFF PAGE REFERENCE REPORT - \PADSPWR\LOGICFILES\filename.SCH - date
time

Signal name

Sheet no. X Y coordinate   Sheet no. X Y coordinate

Signal name

Sheet no. X Y coordinate   Sheet no. X Y coordinate
```

Bill of Materials

The Bill of Materials report produces a user-configurable list of the parts contained in the current schematic. You can direct any part attribute in the schematic to a Bill of Materials report.

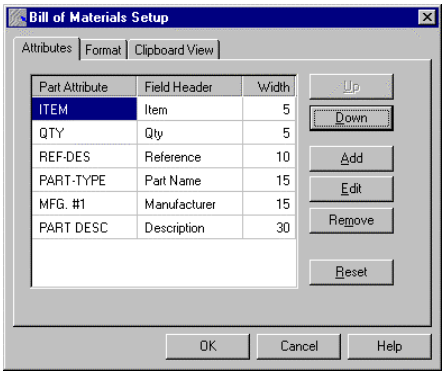
1. In the Reports dialog box, choose Setup to specify or modify the Bill of Materials report characteristics.
2. Choose the tab that represents the Bill of Materials report characteristics that you want to set. Choose from Attributes, Format, or Clipboard View.
3. When you choose one these tabs, the options for that tab appear. Set the options you want to change. Optionally, you can choose the other tabs to set other options if necessary.
4. When you finish the Bill of Materials setup, choose OK. Choose Cancel to close the dialog box without saving any of the changes.

Attributes Tab

The Attribute tab dialog enables you to modify the attribute content, the corresponding column headings, and column width of the Bill of Material report.

The content of the multicolumn list box originates from the .ini file. To update the ini section with new values from the Attributes tab, choose OK.

The attribute order in the content list determines the column arrangement in the BOM report. There is a limit of 12 attributes in the Bill of Materials report.



Part Attribute

Lists up to twelve attribute names. The name must exactly match the exact name of the attribute in the part library. Each attribute defines a column in the BOM report.

Field Header

Specifies the column heading for each attribute in the report. You can specify any character except the colon (:).

Width

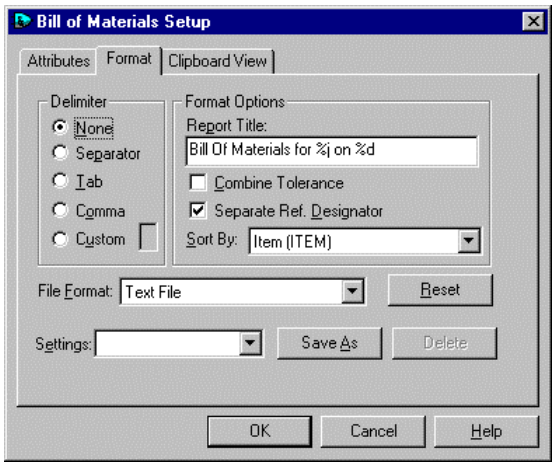
Specifies an integer that represents the number of characters across the column for each attribute in the report. Type a value between 1 and 200. Do not exceed the total report width of 200.

PowerLogic reserves a space to separate columns. Therefore, the actual column width is one character less than the specified number.

If the attribute length exceeds the column width, the attribute name word wraps within that column.

Format Tab

The Format tab enables you to modify the output format of the Bill of Material report.



None

Uses no delimiter character to distinguish BOM report columns.

Separator

Uses a vertical bar between BOM report fields.

Tab

Uses tabs between BOM report fields.

Comma

Uses a comma between BOM report fields.

Custom

Allows you to specify any character as a delimiter between BOM report fields.

Report Title

Creates a custom header for the report. The variable %j is replaced by the filename and %d is replaced by the date.

Combine Tolerance

Combines the Value and Tolerance attributes of a part in the part name. For example, the ¼ watt resistor would have a part type name of R1/4W or R1/4W.4.7K,+/-5%. PowerLogic evaluates parts that have either a different Value or Tolerance attribute as different part types.

Separate Reference Designator

Combines into a single report entry any parts that have identical attribute values that only differ by their reference designators. This reduces the BOM report size. For example, if U1 through U10, U12, and U16 to U18 are all identical parts, then PowerLogic generates one entry with the reference designator column containing the string: U1-10 U12 U16-18. When turned on, parts always appear on separate lines.

Sort By

Sorts the report list by the specified attribute. If the sort field is off, parts are sorted by Part Type name.

File Format

Specifies the output file format.

Settings

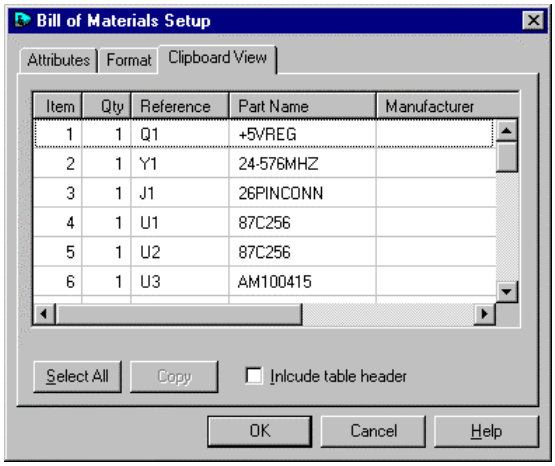
Saves BOM format settings for the current design to a specified file, allowing you to create different format configurations for different designs. When you finish setting the format, choose Save and type a filename in the Save Scheme dialog box. PowerLogic saves BOM format settings files in the \padspwr\powerlogic directory with a .bom extension. Choose Delete to remove the selected setting file from the drop-down list box.

Clipboard View Tab

The Clipboard View allows you to preview the Bill of Materials report format and copy any selected lines of the file to a Windows Clipboard.

The default view orders the attributes by a sort field you specify in the Format tab dialog. To sort rows by another attribute, place the cursor in the column header area you want to sort by and click the left mouse button. To sort the same column in reverse order, click the left mouse button on the column header again.

To change column width, place the cursor on the column line in the heading area. When the arrow cursor changes shape, click and drag the left mouse button in either direction to modify the width. Changing column width on the Clipboard View tab does not change the width values on the Attributes tab.



Select All

Selects all rows in the BOM report list.

Copy

Copies selected rows in the BOM to the Clipboard. You can then paste the Clipboard contents to another application, such as Microsoft Word. Tabs separate the report list fields into columns.

Include Table Header

Includes the column header information (Item, Qty, etc.) in the report.

Index

–2–

2D lines
  combine items, 8-7
  creating, 8-4
  modifying, 8-5
  outputting, 2-4
2D line item, 8-8

–A–

About command, 1-12
Adding
  attributes to libraries, 7-22
  bus, 4-13, 4-14
  connection, 4-2, 4-5
  connector pins, 4-5
  from library, 8-2, 8-7
  gate, 7-18
  item, 1-27
  new attribute to library, 6-9
  new attributes, 5-4
  off-page references, 4-6
  part, 4-2
  part, ECO file format, 2-16
  pin, 7-18
  pin to net, ECO file format, 2-16
  polygons, 8-4
  power and ground connections, 4-7
  terminal, 7-35, 7-36
  text, 8-1, 8-2
  to library, 8-8
Advanced pen plotter setup, 11-7
Advanced photoplotter setup, 11-9
Advanced scripts, 9-24
Against rule object, 10-8
Allow ripup, 10-14
Allow shove, 10-14
Alphanumeric pins tab
  (part information dialog box), 7-24
Alternate decal assignments, 7-17
Alternate for ground part, 7-43
Alternate for off-page, 7-41
Alternate for power part, 7-42
Alternate symbols, 4-5
Angles, modeless commands, 1-22
Annotation, forward and backward, 2-8
ANSII symbol standards, 4-5
Apertures, 11-8
  count, 11-9
Apply, xx
Arcs, pulling, 8-6
.asc files, importing, 2-2
ASCII, 2-4
  exporting, 2-3
  file format, 2-5
  importing, 2-2
Assign nets, 10-21
Assigning signal names to pins, 7-21
Attribute manager, 1-10
Attributes
  adding new, 5-4
  adding new to library, 6-9
  adding to a schematic, 5-3
  adding to library, 7-22
  browse in library, 5-1, 6-10
  compare for ECO, 2-12
  creating, 5-1, 5-2
  creating in library, 6-6
  deleting from a schematic, 5-3
  labels, creating, 5-9
  labels, overview, 5-8

managing in library, 5-1, 6-7
managing in schematic, 5-2
modifying, 5-1, 5-4
query/modify for parts, 5-5
Attributes tab
(bill of materials setup dialog box), 11-17
Attributes tab
(part information dialog box), 7-22
Auto route, 10-14
Automatically updating OLE links, 9-10
Automation features in PowerLogic, 9-1, 9-17
Automation server, 9-3
Automation tools, 9-13
Autorouter, 10-14

–B–

Backward annotation, 2-8
attribute level, 2-15
from PowerPCB, 1-10, 2-12
gate level, 2-14
net level, 2-15
part level, 2-13
pin level, 2-14
Base name, 4-14
Biasing layers, 10-14
Bill of materials report, 11-17
attributes tab (bill of materials setup dialog box), 11-17
clipboard view tab (bill of materials setup dialog box), 11-20
format tab (bill of materials setup dialog box), 11-18
Bit range, 4-14
Bitmap, copying as, 1-6, 4-26
BLT modeless command, 1-23
BMW modeless command, 1-23
.bom file, 11-19
Breaking OLE links, 9-9
Browse library attributes, 5-1, 6-10
Bus
adding, 4-13
adding connections to, 4-14
deleting a segment, 4-15
extending, 4-15
modifying, 1-29
moving segment, 4-15
naming, 4-14
query/modify name, 4-34
splitting, 4-15
Bus name
query/modify, 4-34
Bus toolbox, 1-17, 4-13
adding bus, 4-14
adding connections to a bus, 4-14
deleting a bus segment, 4-15
extending a bus, 4-15
moving a bus segment, 4-15
naming a bus, 4-14
splitting a bus, 4-15
viewing, 1-11, 1-13

–C–

CAE decal, 6-2
creating, 7-9
editing, 7-1
Cancel, xx
Capacitance, 10-17
Capture view, 1-9, 1-21
.ccf file, 3-8
Change pin decal, 7-35
Change source dialog box, 9-9
Changing a connection, 4-8
Changing an OLE icon, 9-5
Changing library parts, 7-2
Changing the part type, 4-28
Character limit, 4-8
Check boxes, xix
Check part, 7-46
Checking
cross-talk conditions, 10-8
electrodynamic, 10-15
parallelism, 10-8
tandem, 10-8
Circles
adding, 8-4

Circular interpolation, 11-10
Class rules, 10-2, 10-4
  adding class, 10-4
  class name, 10-4
  clearance, 10-4, 10-5
  deleting, 10-4
  high-speed, 10-4, 10-5
  nets, 10-5
  renaming, 10-5
  report, 10-10
  routing, 10-4, 10-5
Clearance rules, 10-1, 10-11
  class, 10-4, 10-5
  default, 10-3
  net, 10-6
  rule sets, 10-8
Clearance values
  body to body, 10-12
  drill to drill, 10-12
  edge-to-edge, 10-12
  pad to corner, 10-12
  SMD to corner, 10-12
  SMD to via, 10-12
  via to via, 10-12
Clipboard view tab (bill of materials setup dialog box), 11-20
Colors
  background for OLE object, 9-7
  configurations, 3-8
  in Visual Basic editor, 9-18
  setting, 3-7
  to print or plot, 11-4
Combine, 8-2, 8-7
Command button, xix
Command line options
  start-up options, 1-1, 1-2
Comparing netlists, 9-15
Comparing PowerLogic schematics with PowerPCB designs, 2-8
Compress mouse mode, 3-15
Conditional rules, 10-7, 10-16
Connect command, 1-16
Connect to PCB, OLE, 9-14
Connection properties, 1-16
Connections
  adding, 4-2, 4-5, 4-6
  adding off-page references, 4-6
  adding power and ground, 4-7
  adding to a bus, 4-14
  changing, 4-8
  moving, 4-20
  naming, 4-8
  outputting, 2-4
  power and ground, 4-7
  removing from group, 4-22
  removing outside boundary of group, 4-22
  splitting, 4-21
Connector pins, 4-5
  adding, 4-5
Connector tab (part information dialog box), 7-25
Connectors
  creating new, 7-28
  editing, 7-25
Contents command, 1-12
Control key shortcuts, 1-25
Conventions
  design, xvii
  documentation, xviii
  Windows, xix
Converting an OLE object, 9-1, 9-10, 9-11
Coordinate mode, 3-14, 3-15
Coordinates, 1-4
Copper sharing, 10-14
Copper thickness, 10-20
Copy, 1-6, 4-26
  as bitmap, 1-6, 4-26
  group to file, 4-25
  hierarchical symbol, 4-18
  OLE objects, 9-8
  to file, 1-6
Copy mode, 4-18
Copy mode icon, 1-13
Creating
  2D line, 8-1, 8-4
  attributes, 5-1, 5-2

attributes in the library, 6-6
bottom-up hierarchy, 4-11
CAE decals, 7-9
connectors, 7-28
hierarchical symbols, 4-9
labels, 5-9
media from current session, 1-24
media from previous session, 1-25
multigate parts, 7-7
netlists, 2-5
new symbols from existing symbols, 7-44
parts, 7-26
PCB file, 9-16
pin decals, 7-30
reports, 11-11
top-down hierarchy, 4-10
single gate parts, 7-2
special symbols, 7-40
Visual Basic scripts, 9-19
Cross-probing, 9-13, 9-15
Cross-talk conditions checking, 10-8
Customizing plots, 11-2
Customizing PowerLogic, 3-1, 3-9
Cut, 1-6, 4-26
OLE objects, 9-8

–D–

Data types, library, 6-2
Database minimum, xvii
D-code, 11-8, 11-9
Decal
alternate assignments, 7-19
filtering, 7-15
gates, 7-17
alternate assignments, 7-17
gate and pin swap, 7-17
outputting, 2-4
Decal assignments
query/modify for PCB, 4-31
Decal parameters
hierarchical symbol, 4-9
Decal wizard, 7-34
Default rules, 10-2, 10-3
clearance, 10-3
high-speed, 10-3
routing, 10-3
default.txt, 3-1, 3-9
Delay, 10-17
Delete command, 1-7
Deleting
all OLE objects command, 1-8
bus segments, 4-13
gates, 7-18
groups, 4-24
high-speed rules, 10-17
library objects, 6-4
OLE objects, 9-13
parts, ECO file format, 2-17
pin from net, ECO file format, 2-17
symbols, 4-21
Delete mode, 4-21
Delete mode icon, 1-13
DeMorgan representation, 4-5
Design and hierarchy modeless commands, 1-22
Design conventions, xvii
Design grid, 1-4, 1-18
Design operations, 4-1
overview, 4-1
Design preferences, 3-4
Design rules, 1-9, 10-1
class, 10-2, 10-4
clearance, 10-1, 10-11
conditional, 10-7, 10-16
default, 10-2, 10-3
dialog box, 10-1
differential pairs, 10-9
exporting to PowerPCB, 9-15, 10-18
hierarchy, 10-2, 10-3
high-speed, 10-1, 10-15
importing from PowerPCB, 9-16, 10-18
outputting, 2-4
net, 10-2, 10-6
report, 10-10
routing, 10-1, 10-12
setting, 10-1

Design toolbox, 1-17, 4-1
add connection, 4-5
add part, 4-2
displaying, 1-13
modify hierarchical symbol, 4-12
new hierarchical symbol, 4-9
swap pins, 4-13
swap reference designator, 4-12
viewing, 1-11
Differences report, ECO, 2-11
Differential pairs, 10-9
Directories, 1-25
Disconnect from PCB, OLE, 9-14
Display colors, 1-9
Display grid, 1-18
Displaying OLE objects, 9-6
Docking toolboxes, 1-17
Documentation conventions, xviii
Documentation layers, 10-19
Drafting items
adding from library, 8-7, 8-8
adding to library, 8-8
query/modify, 8-9
Drafting objects, modeless commands, 1-23
Drafting operations, 8-1
Drafting toolbox, 1-17, 7-34, 8-1
add drafting item from library, 8-7
add text, 8-2
add drafting item to library, 8-8
combine, 8-7
create 2D line, 8-4
displaying, 1-13
modify 2D line, 8-5
viewing, 1-11
Drop-down list box, xix

–E–

ECO
changes, included items, 2-9
creating on ECO file, 2-9
differences report, 2-11
file format, 2-16
forward and backward annotation, 2-8
ECO operations, 2-8
ECOGEN, 9-15
EDC, 10-15
Edge-to-edge clearance values, 10-12
Edit menu, 1-5
Edit operations, library, 6-3
Edit part/hierarchical symbol, 4-2
Editing, 4-26, 8-8
CAE decals, 7-1
connectors, 7-25
library objects, 6-3
logic families, 7-13
off-page symbol sheet numbers per line, 4-35
OLE links, 9-8
OLE objects, 9-11
part types, 7-1
PCB decals, 7-1
text, 8-8
Editor, Visual Basic, 9-17
Electrical layers, 10-19, 10-20
adding, 10-22
inner, 10-19
reassigning layer data, 10-22
Electrodynamic checking, 10-15
Embedded objects, 9-2
Exit command, 1-5
Export, 1-5, 2-3
ASCII, 2-3
design rules to PowerPCB, 1-10, 9-15
library, 6-4
OLE, 2-3
rules to PCB, 10-18
Extend bus, 4-13, 4-15
Extents, 1-9

–F–

File format for ECO, 2-16
File menu, 1-4
File operations
creating a new, 2-1
importing and exporting, 2-2
opening, 2-1
saving, 2-2

Files
ASCII, 2-4
copying a group to, 4-25
copying to, 1-6
creating new, 2-1
exporting, 2-3
importing, 2-2
macro, 3-11
opening, 2-1
saving, 2-2
Filter, 1-7
decals, 7-15
gates, 1-15
nets, 1-15
parts, 1-14
pins, 1-15
Floating toolbars, 1-17
Footprints, 7-14
Format tab (bill of materials setup dialog box), 11-18
Forward annotation, 2-8
Forward ECO to PCB, 1-10, 2-8
Free text, 8-2
Frequently asked questions, xxi, 1-12
Function key shortcuts, 1-26
future.dat, 3-10
Futurenet, 3-10

–G–

Gap, 10-16
Gates
add, 7-18
delete, 7-18
swapping, 7-17
Gates filter, 1-15
Gates tab
(part information dialog box), 7-16
General tab
(part information dialog box), 7-11
Get gate decal from library dialog box, 7-32
Get part type from library dialog box, 7-33
Global preferences, 3-2
Grayed text, xix
Grid settings and work area, 1-18
Grid settings, modeless commands, 1-22
Group operations, 4-22
adding to, 4-22
copying, 1-7, 4-23
copying to a file, 4-25
deleting, 4-24
displaying during move or copy, 4-24
drawing, 4-24
erasing, 4-24
moving, 4-23
pasting from a file, 4-25
preserving reference designators, 4-25
removing connections, 4-22
removing connections outside the boundary of, 4-22
removing from, 4-22
setting a new origin for, 4-22
Group select mode, 1-14, 4-17
.grp file, 1-6
GUI descriptions, 1-3

–H–

Heights/widths of text, 8-3
Heights/widths preferences, 3-6
Help, xx
Help menu, 1-12
HGML, 11-7
Hierarchical component
query/modify, 4-34
Hierarchical design, 4-8
creating bottom-up, 4-11
creating top-down, 4-10
moving up, 4-12
pushing into, 4-12
rules hierarchy, 10-2
Hierarchical symbol wizard, 4-9
Hierarchical symbols
copying, 4-18
creating, 4-9
deleting, 4-21
modifying, 4-12
Hierarchy of design rules, 10-2, 10-3

High-speed rules, 10-1, 10-15
class, 10-4, 10-5
deleting, 10-17
net, 10-6
rule sets, 10-8
How to use command, 1-12
HPGL, 11-7

–I–

IEEE symbol standards, 4-5
Impedance, 10-17
Import, 1-5
ASCII, 2-2
design rules from PowerPCB, 1-10, 9-16, 10-18
library, 6-4
OLE, 2-3
Include design rules, netlist output, 2-7
Include part attributes, netlist output, 2-7
Include subsheets, netlist output, 2-6
Include value and tolerance, netlist output, 2-7
Inner electrical layers, 10-19
Insert new OLE objects, 1-8, 9-3
Insert OLE objects from file, 9-4
Insert PowerPCB objects, 1-8
Interface, 1-3

–J–

Job limits, 11-14
Join two nets together, ECO file format, 2-17
Justification, 8-3, 11-5

–L–

Labels, 5-8
creating, 5-9
Layers
adding electrical, 10-22
biasing, 10-14
definition, 1-9, 10-19
documentation, 10-19
inner, 10-19
plane, 10-21
reassigning, 10-22
setup, 10-19, 10-21
thickness, 10-20
Length matching, 10-17
Length minimization, 10-13
Library, 1-5, 6-1
adding drafting items from, 8-7
adding drafting items to, 8-8
adding new attributes to, 6-9, 7-22
browsing attributes, 5-1, 6-10
changing ports, 7-2
creating attributes in, 6-6
data types, 6-2
deleting objects, 6-4
editing objects, 6-3
edit operations, 6-3
exporting, 6-4
importing, 6-4
list, 6-2
managing attributes, 5-1, 6-7
managing special symbols in, 7-46
modifying the list order, 6-4
new, 6-2
new objects, 6-3
operations, 6-1
selecting, 6-2
updating parts, 7-2
updating pin from, 7-45
updating symbol from, 7-45
Library list, 6-2
modifying, 6-4
Library manager, 6-1, 6-3, 7-10
Library part attributes, outputting, 2-4
Limits, job, 11-14
Line width setting, 1-4
Lines, 6-2
modifying, 8-1, 8-5
outputting, 2-4
Linked objects, 9-2
Linking and embedding, 9-1. *See* OLE
Links command, 1-8
List box, xix

.log file, 3-18
Log test, 1-1
  dialog box, 1-25
Logic families, 7-12
  adding, 7-13
  deleting, 7-13
  dialog box, 7-13
  editing, 7-13
  prefix, 7-13
logic00.sch, 3-4

–M–

Macros, 1-11, 3-10, 3-11
  and dialog boxes, 3-11
  assigning shortcut keys, 3-13, 3-16
  coordinate mode, 3-14
  creating, 3-12, 3-14
  dialog box, 3-12
  editing, 3-13
  file format, 3-12
  files, 3-11
  macro status dialog box, 3-16
  mouse compression mode, 3-14
  overview, 3-10
  playing back, 3-17
  record prompt mode, 3-14
  recording, 3-14
  recording PowerLogic sessions, 3-18
  repeating, 3-13
  running, 3-13
Manage library attributes dialog box, 6-7
Managing schematic attributes, 5-2
Manually updating OLE links, 9-10
.mcr file, 3-11
Media, creating from previous session, 1-1
Media directories, 1-25
Media wizard, 1-1
  dialog box, 1-24
Menu bar, xix, 1-3
  viewing, 1-11
Menu commands, 1-1, 1-4
Menus, 1-3
  pop-up, 1-27
Middle mouse button, 1-19
Mid-driven nets, 10-13
Minimization of nets, 10-13
Modeless commands, 1-1, 1-12, 1-22
Modes of operation, 4-17
  copy, 4-18
  delete, 4-21
  group select, 4-17
  move, 4-20
  object select, 4-21
  verb, 4-17
Modifying
  2D lines, 8-1, 8-5
  attributes, 5-1, 5-4
  busses, 1-29
  hierarchical symbols, 4-12
  library list order, 6-4
  line items, 1-29
  parts, 1-29
Mouse actions defined, xviii
Mouse click substitutes, modeless commands, 1-23
Mouse compression mode, 3-14
Move command, 1-7
Moving
  bus segments, 4-13, 4-15
  connections, 4-20
  groups, 4-23
  OLE objects, 9-8
  text, 4-20
  up, hierarchical design, 4-12
Move mode, 4-20
Move mode icon, 1-13
Multicolumn list box, xix
Multigate parts, 4-4
  creating, 7-7

–N–

Names for busses, 4-14
Names for connections, 4-8
Names for pins, 7-18
Naming conventions for special symbols, 7-45
Net assignments, plane layers, 10-21
Net filter, 1-15

Net rules, 10-2, 10-6
clearance, 10-6
high-speed, 10-6
report, 10-10
routing, 10-6
Net statistics report, 11-14
error messages, 11-14
Netlists
comparison, 9-16
creating, 2-5
include design rules in, 2-7
include part attributes in, 2-7
include subsheets, 2-6
include value and tolerance in, 2-7
sending to PowerPCB, 1-10, 9-16
Netlist to PCB command, 1-10
Netlist to PCB dialog box, 2-6
Netname, 9-17
query/modify, 4-33
Nets
assigning, 10-21
joining through ECO, 2-17
mid-driven, 10-13
minimization, 10-13
renaming, ECO file format, 2-18
splitting, 2-18
New attributes, 5-1
New files, 2-1
New hierarchical symbols, 4-2
New libraries, 6-2
New library objects, 6-3
New OLE objects, 9-3, 9-4
next.ini, 3-18
next.log, 3-18
Next/previous command, 1-16
Numeric keypad, 1-21

–O–

Object browser, 9-20
Object command, 1-8
Object linking and embedding. *See* OLE
Object locator list, 1-16
Object select mode, 4-21
Object selector toolbar, 1-14, 4-22
Objects
embedded, 9-2
linked, 9-2
Off-page references, 4-6
Off-page reports, 11-16
Off-page symbols
sheet numbers per line, 4-35
update from library, 7-45
OK, xx
OLE, 9-1
automation, 9-2
background color of object, 9-7
breaking object link, 9-9
change source dialog box, 9-9
changing an object icon, 9-5
connecting to PCB, 9-14
converting objects, 9-10
copying objects, 9-8
creating new object from file, 9-4
creating new objects, 9-3
cross-probing, 9-13
cutting objects, 9-8
delete all OLE objects command, 1-8
deleting objects, 9-13
disconnecting from PCB, 9-14
displaying objects, 9-5, 9-6
editing commands for, 9-10
editing links, 9-8
exporting, 2-3
exporting OLE files, 2-2
importing, 2-2
importing OLE files, 2-2
inserting new objects, 1-8, 9-3
inserting objects from file, 9-4
inserting PowerPCB object, 1-8
linked and embedded object automatic updates, 9-10
linked and embedded object manual updates, 9-10
linking and embedding, methods of, 9-3
linking inserted and embedded objects, 9-5
links command, 1-8
listing links, 9-8
managing nets, 9-13

managing received selections, 9-13
managing selections to send, 9-13
moving objects, 9-8
overview, 9-1
paste objects, 9-8
PowerPCB connection dialog box, 9-13
printing and plotting objects, 9-13
receiving selections, 9-14
redrawing objects, 9-7
saving objects, 9-12
selecting objects, 9-7
sizing objects, 9-8
turning off object display, 9-6
updating linked and embedded objects, 9-9
visual editing, 9-2, 9-10
.ole file, 9-12
OLE icon, changing, 9-5
OLE import/export, 2-3
OLE PowerPCB connection, 1-11
Open command, 1-4
Open icon, 1-13
Opening
a PCB file, 9-16
Visual Basic scripts, 9-19
Operational modes, 4-17
Option buttons, xix
Options, start-up, 1-2
Orientation of print or plot, 11-5
Origin, 1-18
setting the group origin, 4-22
setting the part origin, 7-46
Output device, 11-2
Output filename, netlist output, 2-6
Output format, 2-4
netlist output, 2-6

–P–

padsnet.asc, 11-11
Palette, 3-8
Parallel checks, 10-8
Parallel rules, 10-16
Parent sheet, 4-34
Part attributes, query/modify, 5-5
Part editor, 1-10, 3-6, 4-11, 4-12
Part editor operations, 6-4, 7-1, 7-2, 7-5, 7-10, 7-27, 7-29
assigning alternates for ground part, 7-43
assigning alternates for off-page, 7-41
assigning alternates for power part, 7-42
changing parts, 7-2
check part, 7-46
creating CAE decals, 7-9
creating connectors, 7-28
creating multigate parts, 7-7
creating parts, 7-26
creating pin decals, 7-30
creating new special symbols from existing symbols, 7-44
creating single gate parts, 7-2
decal wizard, 7-34
get gate decal from library, 7-32
get part type from library dialog box, 7-33
managing special symbols in the library, 7-46
modifying parts, 1-29
overview, 7-1
reviewing parts statistics, 7-11
save part and gate decals, 7-6
saving a modified decal, 7-5
selecting gate decals, 7-8
selecting pin decals, 7-41
setting the part origin, 7-46
special symbol naming conventions, 7-45
special symbols, 7-29, 7-40
terminal toolbox, 7-35
update off-page from library, 7-45
update pins from library, 7-45
Part filter, 1-14
Part information dialog box, 7-10
alphanumeric pins tab, 7-24
attributes tab, 7-22
connector tab, 7-25
gates tab, 7-16
general tab, 7-11
PCB decals tab, 7-14
signal pins tab, 7-21

Part statistics report, 11-13
Part statistics, reviewing, 7-11
Part text visibility dialog box, 4-29
Part text visibility, query/modify, 5-7
Part type, 6-2
  changing, 4-28, 4-30
  editing, 7-1
Parts
  adding, 4-2
  adding multigate, 4-4
  adding single gate, 4-4
  changing from library, 7-2
  modifying, 1-29
  outputting, 2-4
  query/modify, 4-27
  renaming, 4-28
  renaming, ECO file format, 2-18
  updating from library, 7-2
Paste, 1-6, 4-26
Pasting
  from file, 1-7
  group from file, 4-25
  OLE objects, 9-8
PCB decals, 4-29, 6-2
  editing, 7-1
  query/modify assignment, 4-31
PCB decals tab
  (part information dialog box), 7-14
Pen plotter, 11-2
  advanced setup, 11-7
  setup, 11-6
Photoplotter, 11-2
  advanced setup, 11-9
  setup, 11-8
Pins
  adding, 7-18
  assigned to seleted gate, 7-18
  assigning signal names to, 7-21
  changing sequence number, 7-36
  naming, 7-18
  removing, 7-18
  sequence in decal, 7-19
  swap ID, 7-19
  swapping, 4-13, 7-36
  terminal number sequence, 7-18
  type, 7-18
  updating from library, 7-45
Pin decal list management dialog box, 7-31
Pin decals
  changing, 7-35, 7-37
  creating new, 7-30
  managing, 7-31
Pin filter, 1-15
Pin names
  changing, 7-35, 7-38
  setting, 7-35, 7-38
Pin numbers
  changing, 7-35, 7-37
  setting, 7-35, 7-37
Pin sequence numbers, changing, 7-39
Pin swaps, setting, 7-39
Pin types
  changing, 7-38
  setting, 7-35
Plane layer nets dialog box, 10-21
Plane layers, 10-21
  net assignments, 10-21
Playback, macros, 3-17
Playback, recorded PowerLogic session, 1-1
Plot, 11-1
  colors, 11-4
  customizing, 11-2
  device setup, 11-2
  justification, 11-5
  OLE objects, 9-13
  options, 11-5
  orientation, 11-5
  output devices, 11-2
  pen plotter advanced setup, 11-7
  pen plotter setup, 11-6
  photoplotter advanced setup, 11-9
  photoplotter setup, 11-8
  preview, 11-4
  preview selections, 11-2
  scaling, 11-5
Plot dialog box, 11-1
Plot options, 11-5

Plot parameters, outputting, 2-4
Point of origin, 1-3
Polygons, adding, 8-4
Pop hierarchy command, 1-9
Pop-up menus, xix, 1-27
  anything selected, 1-30
  part or gate selected, 1-27
Postage stamp, 1-20
PostScript printing, 11-11
  to a file, 11-11
Power and ground connections, 4-7
PowerLogic
  automation features, 9-17
  colors, setting, 3-7
  customizing, 3-1, 3-9
  database minimum, xvii
  frequently asked questions, xxi
  interface description, xix, 1-3
  menu commands, 1-4
  OLE overview, 9-1
  recording sessions, 3-18
  starting, 1-1
  start-up options, 1-1
  synchronizing with PowerPCB, 9-15
  troubleshooting, 3-18
  Windows conventions, xix
PowerLogic interface, 1-3
PowerPCB connection dialog box, 9-13
Preferences, 1-9, 3-1
  design, 3-4
  global, 3-2
  heights/widths, 3-6
  setting, 3-1
Prefix
  and family, 7-13
  part, 7-12
Preserving reference designators, group, 4-25
Preview
  plot and print selections, 11-2
  print/plot, 11-4
Previous view, 1-9, 1-21
Previous/forward sheet, 1-16
Print, 11-1, 11-2
  OLE objects, 9-13
  orientation, 11-5
  PostScript, 11-11
  setup, 1-5
  to a file, 11-11
  Visual Basic scripts, 9-19
Print manager, 11-11
Print/plot operations. *See* Plot
Priority, 10-14
Pull arc, 8-6
Push hierarchy, 1-9
pwrlogic.ini, 3-18

–Q–

Query mode icon, 1-13
Query/modify, 4-26, 8-8
  bus name, 4-34
  drafting object, 8-9
  hierarchical component, 4-34
  net name, 4-6, 4-33
  part, 4-27
  part attributes, 5-5
  part text visibility, 5-7
  PCB decal assignments, 4-31
  text, 8-8

–R–

Ratsnest, 10-13
Reassigning layers, 10-22
Record mode toolbox, 3-14
Record prompt mode, 3-14, 3-15
Rectangles, adding, 8-5
Redo, 1-6, 1-13
Redraw, 1-8, 1-14
Redrawing OLE objects, 9-7
Reference designators
  preserving in group, 4-25
  swapping, 4-12
Renaming a net, ECO file format, 2-18
Renaming part, 4-28
  ECO file format, 2-18

Reports, 1-5, 11-11
bill of materials, 11-17
classrules, 10-10
creating, 11-11
net statistics, 11-14
netrules, 10-10
off-page, 11-16
output, 10-11
part statistics, 11-13
rules, 10-10
unused gates and pins, 11-13
report.rep, 11-11
Resizing toolboxes, 1-17
Restoring views, 1-21
RGL reports, 9-22
Ripup, allowing, 10-14
Rotating text, 8-3
Routing options, 10-14
Routing rules, 10-1, 10-12
class, 10-4, 10-5
default, 10-3
deleting, 10-15
net, 10-6
Rules, 10-2, 10-3. *See* Design rules
Rules dialog box, 10-1
Rules report, 10-10

–S–

Samples
automation, 9-18
Visual Basic scripts, 9-23
Save, 1-4
Saving
modified decal with a different name, 7-5
modified part type with a different name, 7-5
offpage to library, 1-10
part and gate decals, 7-6
dialog box, 7-6
to library, 8-2
views, 1-21
Save As, 1-5
Save icon, 1-13
Sax Basic Engine dialog box, 9-18
Scaling plots, 11-5
Schematic editor mode, 1-14
Schematic maximum size, xvii
Schematic parameters, outputting, 2-4
Schematics, creating, 4-1
Scripting in Visual Basic, 9-17, 9-24
Scroll bars, 1-19
Search, 1-12
Search modeless commands, 1-22
Segments
move bus segment, 4-15
splitting, 8-6
Select
gate decals, 7-8
dialog box, 7-8
libraries, 6-2
OLE objects, 9-7
pin decals, 7-41
sheets, netlist output, 2-6
Select icon, 1-13
Select items dialog box, 11-3
Selection commands, pop-up menu, 1-30
Sending a netlist to PowerPCB, 9-16
Set verb mode, 1-31
Setting preferences, 3-1
Setting the part origin, 7-46
Setup menu, 1-9
Setup, plotter or printer, 11-2
Setup via dialog box, 10-15
Shapes, adding, 8-4
Sheet command, 1-8, 1-14
Sheet dialog box, 4-16
Sheet list box, 1-13
Sheet number, 11-16
Sheet parameters, outputting, 2-4
Sheets, 1-19
off-page numbers per line, 4-35
parent, 4-34
previous/forward, 1-16
Sheets to print or plot, 11-3
Shielding, 10-17

Shortcut keys, 1-25
assigning to macros, 3-16
Shoving, allowing, 10-14
Show current statement, Visual Basic, 9-20
Signal names, assigning to pins, 7-21
Signal pins, 4-29
Signal pins tab
(part information dialog box), 7-21
Single gate parts, 4-4
creating, 7-2
Sizing OLE objects, 9-8
Source application, opening an OLE object in, 9-11
Source rule object, 10-7
Special symbols, 7-29
creating, 7-40
creating from existing, 7-44
managing in the library, 7-46
naming conventions, 7-45
Splitting
busses, 4-13, 4-15
connections, 4-21
nets, 2-18
ECO file format, 2-18
segments, 4-21, 8-6
Standard toolbar, 1-12
Starting PowerLogic, 1-1
Start-up options, 1-1, 1-2
Statistics
net, 11-14
part, 11-13
Status bar, 1-4
viewing, 1-11
Status window, 1-19
viewing, 1-11
Step and repeat, 4-18
Stub length, 10-17
Suffix, part, 7-12
Swap ID, 7-19
Swapping
gates, 7-9, 7-16
ECO file format, 2-18
pins, 4-2, 4-13, 7-36
ECO file format, 2-18
reference designators, 4-2, 4-12
Switches, command line, 1-1
Symbols
alternate, 4-5
ANSII standards, 4-5
creating hierarchical, 4-9
creating from
deleting hierarchical, 4-21
IEEE standards, 4-5
modifying hierarchical, 4-12
Synchronizing PowerLogic and PowerPCB, 9-15
System status indicator, 1-3

–T–

Tandem checks, 10-8
Tandem rules, 10-16
Terminal number sequence, 7-18
Terminal toolbox, 1-17, 7-35
Terminals, adding, 7-36
Text
adding, 8-2
editing, 8-8
height and width, 8-3
justification, 8-3
moving, 4-20
outputting, 2-4
part text visibility dialog box, 4-29
query/modify, 8-8
query/modify part text visibility, 5-7
rotation, 8-3
Text box, xix
Thermal relief connection pads, 10-21
Thickness of layers, 10-20
Tie dot, 4-6
Toolbar, xix, 1-1, 1-3, 1-12
docking, 1-17
floating, 1-17
object selector, 1-14, 4-22
standard, 1-12
viewing, 1-11

Toolboxes, xix, 1-12, 1-17
bus, 4-13
design toolbox, 4-1
docking, 1-17
drafting, 7-34, 8-1
floating, 1-17
resizing, 1-17
terminal, 7-35
Tools menu, 1-10
ToolTips, 1-17
Trace width, 10-11
Troubleshooting, 3-18
frequently asked questions, xxi, 1-12
Two dimensional lines. *see* 2D line
.txt file, 2-3

–U–

Undo, 1-5, 1-13
Units, plotting, 11-10
Unused gates and pins report, 11-13
Unused pins net
ignoring, 9-17
managing, 9-16
specifying the name of, 9-17
Update off-page from library, 1-10, 7-45
Update pins from library, 1-10, 7-45
Updating library parts, 7-2
Updating linked and embedded OLE objects, 9-9, 9-10

–V–

Verb mode, 4-17
setting, 1-31
Vias, restricting use, 10-14
Views, 1-19
capture, 1-21
controlling, 1-18
name, 1-21
previous, 1-21
restoring, 1-21
saving, 1-21
View menu, 1-8
Visual Basic Editor, 9-17
creating scripts, 9-19
sample scripts, 9-22
advanced, 9-24
Sax Basic Engine, 9-18
scripting, 9-17
Visual Basic scripts, 1-11
accessing, 9-21
adding to a menu, 9-21
creating, 9-19
editing, 9-21
loading, 9-21
opening, 9-19
running, 9-21
samples, 9-22
advanced, 9-24
Visual editing, OLE, 9-2, 9-10

–W–

width.dat, 3-9
Window menu, 1-11
Windows print manager, 11-11
Work area, 1-4
and grid settings, 1-18
dimensions, 1-18

–X–

X,Y coordinate location, 11-16

–Z–

Zero suppress, 11-10
Zoom, 1-8, 1-14, 1-19